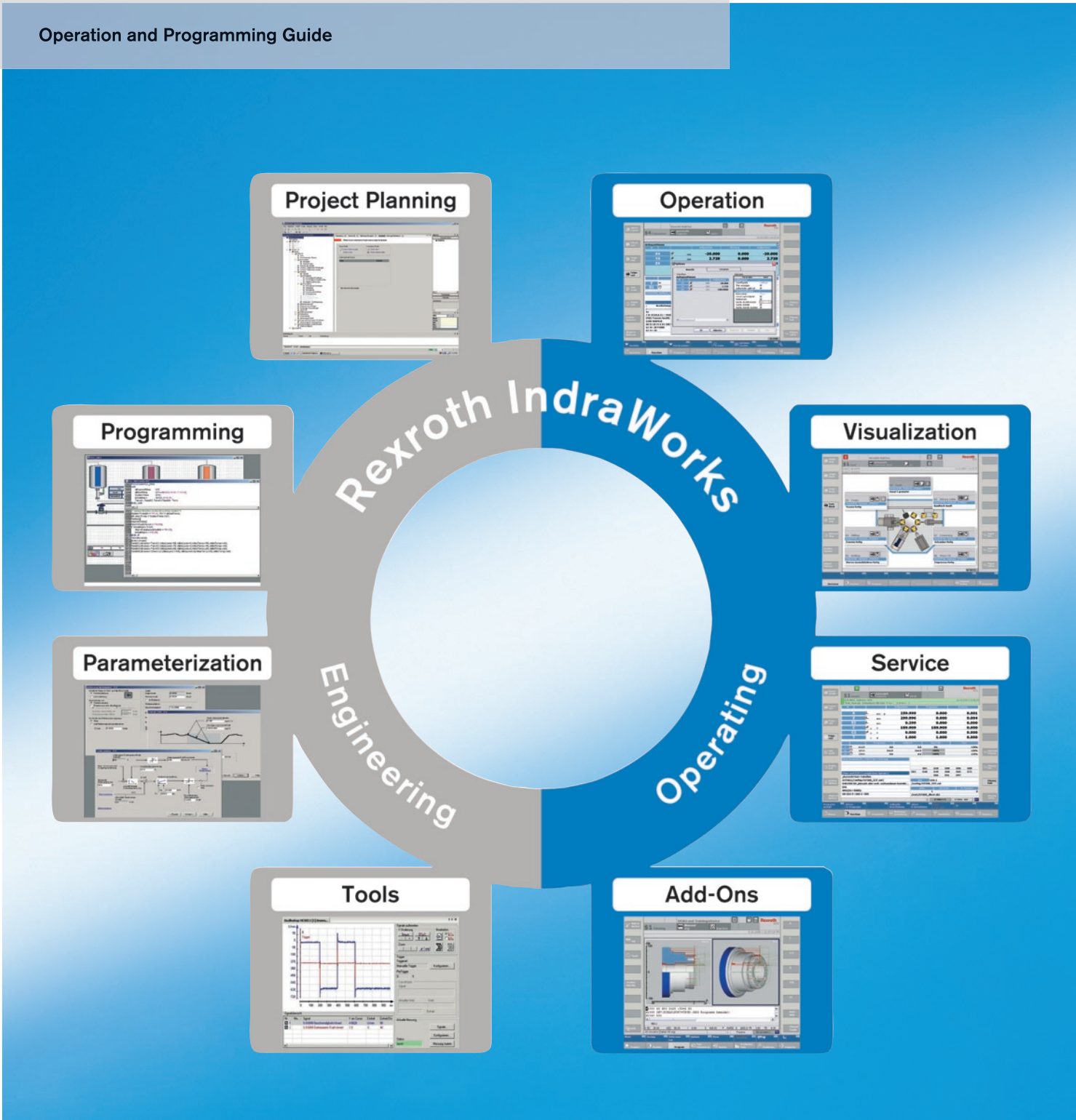
Electric Drives and Controls | Hydraulics | Linear Motion and Assembly Technologies | Pneumatics | Service

Rexroth Bosch Group

Rexroth IndraWorks Engineering 07VRS

R911320741
Edition 01

Operation and Programming Guide



Title Rexroth IndraWorks
Engineering 07VRS

Type of Documentation Operation and Programming Guide

Document Typecode DOK-IWORKS-ENGINEE*V07-AW01-EN-P

Internal File Reference RS-f6aae24c7173a3640a6846a001e2db63-1-en-US-3

Record of Revision

Edition	Release Date	Notes
120-2700-B310-01/EN	06/2007	First Issue V07

Published by Bosch Rexroth AG
Bgm.-Dr.-Nebel-Str. 2 ■ D-97816 Lohr a. Main
Telefon +49 (0)93 52/ 40-0 ■ Fax +49 (0)93 52/ 40-48 85
http://www.boschrexroth.com/
Dept. BRC/EAC4 (JaWa)

Note This document has been printed on chlorine-free bleached paper.

Table of Contents

Table of Contents

Table of Contents

Page

Table of Contents

Table of Contents

Page

Table of Contents

1 Basics

1.1 General Information

IndraWorks is the carrier system for integration of the Bosch Rexroth engineering tools.

Cross-sectional functions, such as project navigation and project administration as well as the generation of project and configuration data, are executed in a centralized manner.

Basically, there are two types of integration:

- When full integration is used, all operator actions are performed in the main window of IndraWorks.

  Examples include parameterization of units in an IndraWorks project or configuration of HMI control elements for visualization panels.
- The connection to IndraWorks allows comfortable integration of 3rd party tools. 3rd party tools can be called directly from the IndraWorks project management in their own display format.

  This type of integration is, for example, used for PLC programming and HMI image configuration.

1.2 About this Manual

1.2.1 General Information

This manual contains information about:

chapter 1.2.2 "Elements of the IndraWorks User Interface " on page 2

provides an overview of the visual components of IndraWorks and explains their functions.

chapter 1.3 "Getting Started" on page 5

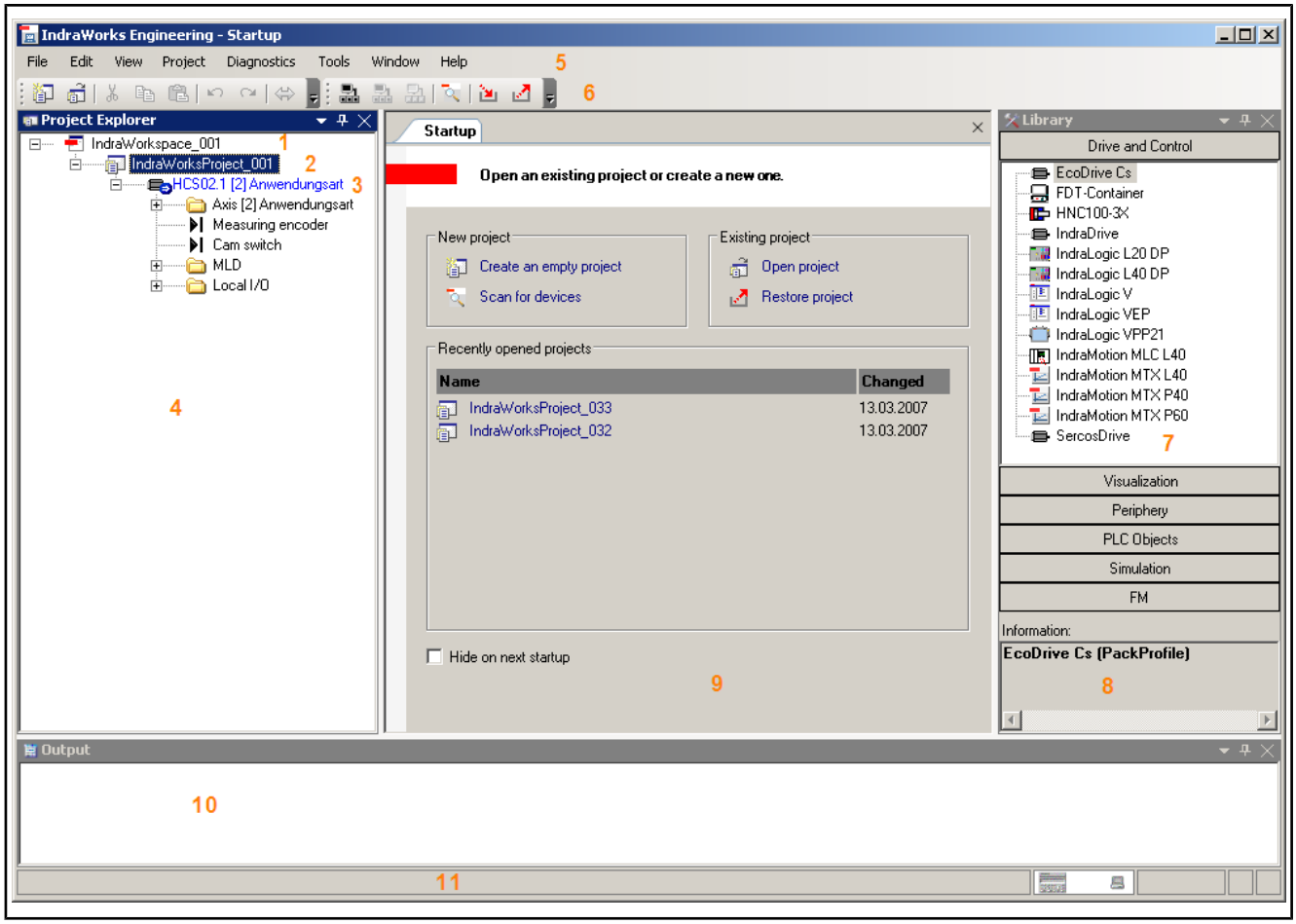
provides instructions and templates for facilitating the use of IndraWorks.

chapter 2.1 "General Information " on page 9

contains a detailed description of the most important parts of IndraWorks.

Basics

1.2.2 Elements of the IndraWorks User Interface



1	Workspace
2	Project
3	Device
4	Project Explorer
5	Menu
6	Toolbars
7	Library
8	Library info
9	Working area
10	Output window and diagnoses
11	Status bar

Fig. 1-1: IndraWorks user interface

1.2.3 Title, Menu and Status Bar

The main window of IndraWorks is enclosed by the title bar and menu bar as well as the status bar.

The title bar shows the name of the window that is active in the working area.

The menu bar contains the menu entries with the corresponding commands. Select a command to execute an action.

The status bar provides information on the current project and on the menu commands.

1.2.4 Toolbars

Toolbars permit quick access to frequently used menu entries and buttons.

You can create your own toolbars and add menus and buttons. The new toolbars will then appear in **View ▸ Toolbars**, where you can activate or deactivate them.

If you exit IndraWorks, the modifications to the toolbars and all new toolbars will be saved. The last settings will be activated when IndraWorks is started the next time.

Standard Toolbar

When IndraWorks is started, the “Standard” toolbar will be shown below the main menu. If necessary, you can move this toolbar with the mouse or hide it by **View ▸ Toolbars**.

Fig. 1-2: Standard toolbar

It provides the following commands:

- Create new project (see main menu **File ▸ New ▸ Project**)
- Open project (see main menu **File ▸ Open ▸ Project**)
- Cut (see main menu **Edit ▸ Cut**)
- Copy (see main menu **Edit ▸ Copy**)
- Paste (see main menu **Edit ▸ Paste**)
- Undo (see main menu **Edit ▸ Undo**)
- Redo (see main menu **Edit ▸ Redo**)
- Synchronize active project
- Toolbars (see main menu **Tools ▸ Customizing ▸ Toolbars**)

1.2.5 working Area

General Information

IndraWorks provides various options and tools supporting you with the management of the windows opened in the working area.

There are two window types in IndraWorks: document windows and tool windows.

Document Windows

IndraWorks supports two types of displaying document windows – tabs and sub windows (MDI - Multiple Documents Interface).

In the sub-window mode, all document windows are arranged in the working area. Several windows can be displayed at the same time (e. g. overlapping).

In the tab mode, only one document window is displayed. It occupies the entire working area. The other open document windows are represented as tabs at the upper edge of the working area. To show such a window, just click on its tab.

To switch between the two modes, select **Windows ▸ Windows as Tabs**.

Windows in the Working Area

Double-click on an object in the project explorer to open a window in the working area. There, you can edit the data or properties of that object. Depending on the object type, the window is a dialog box or an editor. All open document windows are listed in the **Windows** menu. To put a window to the foreground, select the corresponding menu item or click on a visible part of the window in the working area.

Select **Windows ▸ Windows** to open the “Window List” dialog where you can manage the open windows in the working area.

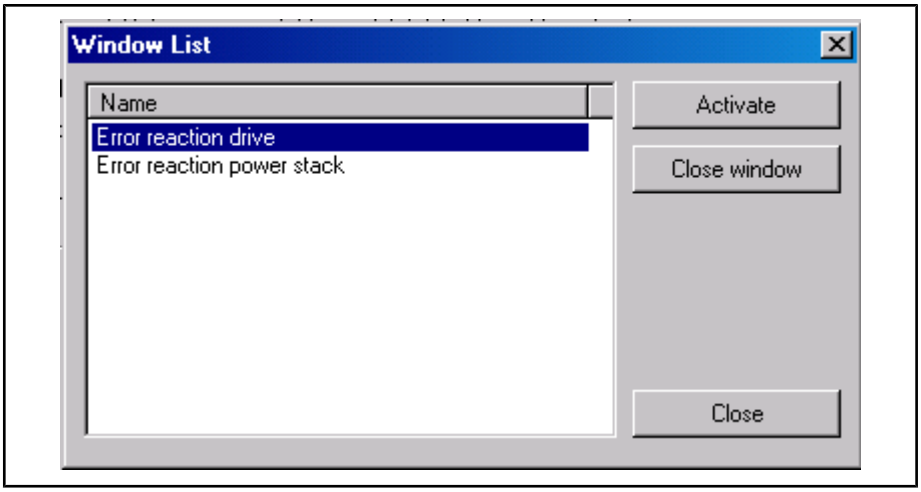


Fig. 1-3: Window List dialog box

Tool Windows

Tool windows are listed in **View** or in **View** ▸ **Other Windows**. To change the behavior of the tool windows, either use the system menu (right-click on the title bar of the tool window) or the **Windows** menu.

Dockable Tool windows are “dockable” by default. A tool window is opened either in the floating mode or it is docked to the edge of the working area. To dock a floating tool window to the edge of the working area, just deselect the “dockable” property. The window will be arranged as an additional tab in the working area. If you reselect the “dockable” property, the window will return to the position it had taken before you arranged it in the working area.

Hide This command hides the active window. You can show the window again using **View** menu.

Floating This command changes the window from floating mode to docked mode and vice versa.

Auto Hide This setting hides the tool windows at the edge of the border such that they are only indicated by a tab displaying the window title. To maximize the window again, just move the mouse pointer over the tab. In this manner, the working area can be enlarged.

Arranging the Windows

Overlapping, Cascading, Tiled In the sub-window mode, the windows in the working area can be arranged in the overlapping, cascading and tiled modes. To achieve this, select the appropriate command from the **Window** menu.

Grouping Document Windows In the tab mode, document windows can be grouped. To achieve this, use the commands of the system menu of the document windows. You can arrange document windows in vertical and horizontal groups and easily move them from one group to the other.

☞ In the tab mode, you can drag and drop document windows within the document window area.

System Menu of Document Windows Using the system menu of the document windows in the sub-window mode, you can minimize, restore, close and move these windows as well as zoom them in and out. Using the system menu in the tab mode, you can create new horizontal and vertical groups, move document windows from one group to the other and close document windows. To activate the document windows one after the other, press <Ctrl>+<F6>.

Full Screen To display as large an area of your document as possible on the screen, activate the full-screen mode by selecting **View ▸ Full Screen**. All tool windows are hidden, and the working area occupies the entire remaining area. The menu bar is still shown. The “Customizing” dialog allows you to add any toolbars. The **View** menu allows you to display any tool window in the full-frame mode. The configuration selected will be stored on exiting the full screen mode and will be restored when it is called the next time. To return to the normal display mode, click on the “Full Screen” button of the “Full Screen” toolbar. Alternatively, you can also press<Ctrl>+<Alt>+<F> or use the menu.

1.2.6 Project Explorer

General Information

The project explorer is arranged to the left of the working area in the IndraWorks main window. The project explorer represents the projects and their components in a structured manner.

Project

A project contains all devices, communication connections and other components required for operating a machine or system. The tree structure of these components reflects the device topology of the automation solution.

Decvice

A device is a component of a project, e. g. a control or a drive. Usually, a device consists of a hardware section and a software section.

In IndraWorks, devices are selected from a library and added to a project. Then the functions available in the devices are called or lower-level devices are projected.

1.2.7 Library Explorer

The library explorer is on the right of the working area in the IndraWorks main window. It represents all libraries available for your projects in a structured manner. Libraries can contain hardware components, i. e. devices, and software components, e. g. function blocks.

1.3 Getting Started

1.3.1 General Information

This section describes some of the typical operating sequences in IndraWorks.

Follow these descriptions, and you will become familiar with the use of IndraWorks and recognize how the various components are cooperating with each other.

1.3.2 Starting IndraWorks

Start IndraWorks via **Start ▸ All Programms ▸ Rexroth ▸ IndraWorks (Version xxx) ▸ Engineering**.

1.3.3 Creating a New Project

To create a new IndraWorks project, select **File ▸ New ▸ Project...**.

The “Create New IndraWorks Project” dialog box will appear. Enter a name for the project and select the directory where the project will be filed.

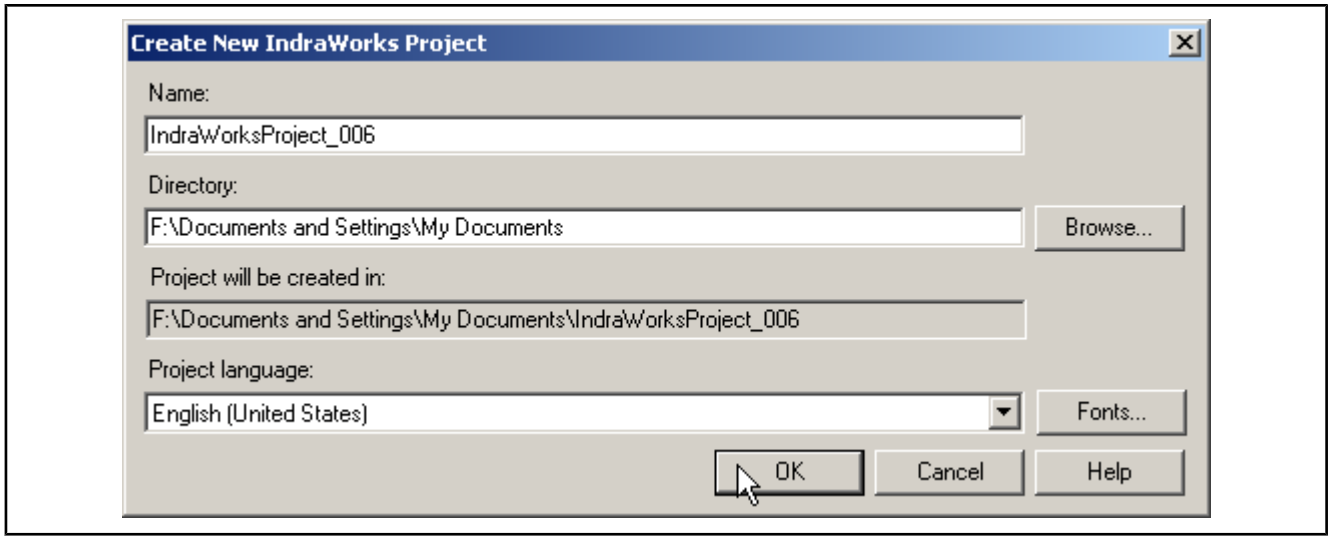


Fig. 1-4: Create New IndraWorks Project dialog box

Click “Browse…” to open the Windows file browser and navigate to the desired project directory. The selected project language will be the master language of your user texts. Define the letter types and font sizes via “Fonts…”.

1.3.4 Opening a Project

To open a project, select **File ▸ Open ▸ Project...**.

Your directory “My Documents” is the default setting in the “Open Project” dialog box. Move to the desired IndraWorks project and confirm your selection with “Open”. The project will now be uploaded.

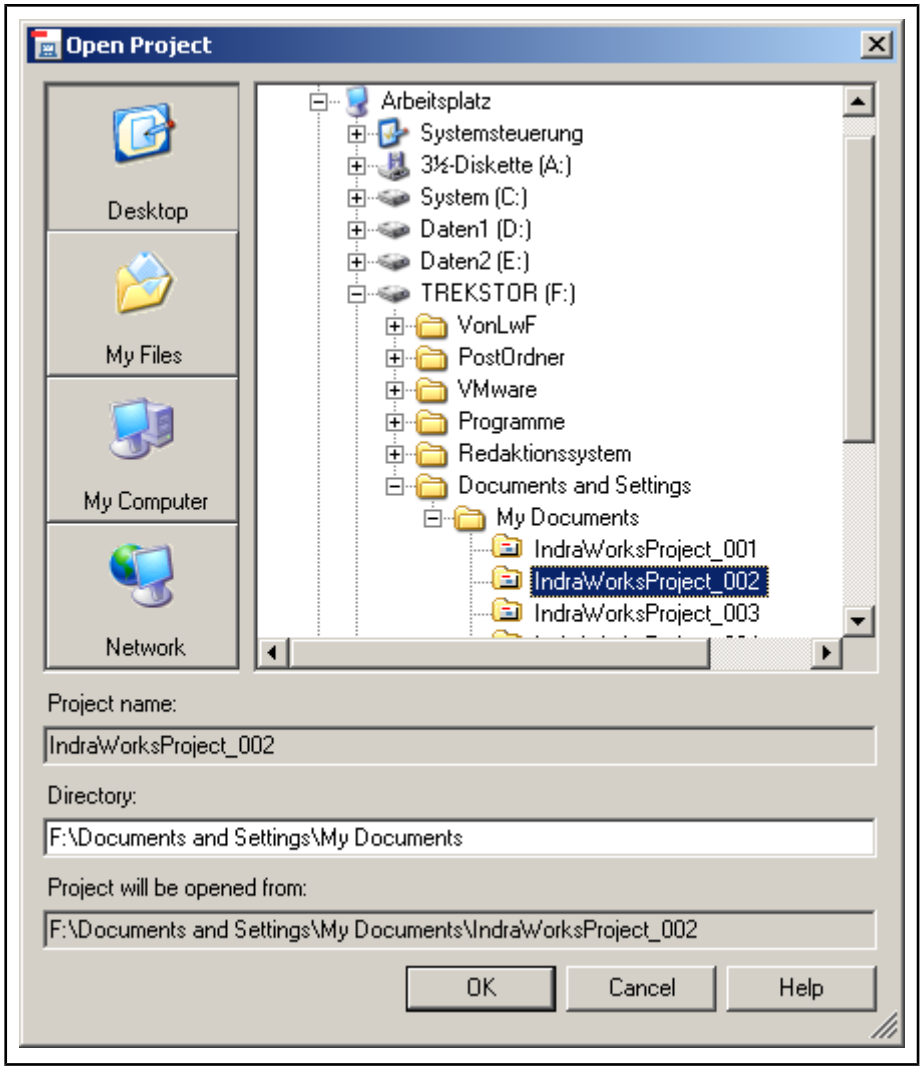


Fig. 1-5: Open Project dialog box

1.3.5 Adding a New Device to a Project

The library explorer displays all available devices of the libraries installed.

To add a device to the current project, drag the device from the library to the project. You can drag the device only directly to the project folder or to devices of the project which accept the currently selected device type as a subdevice.

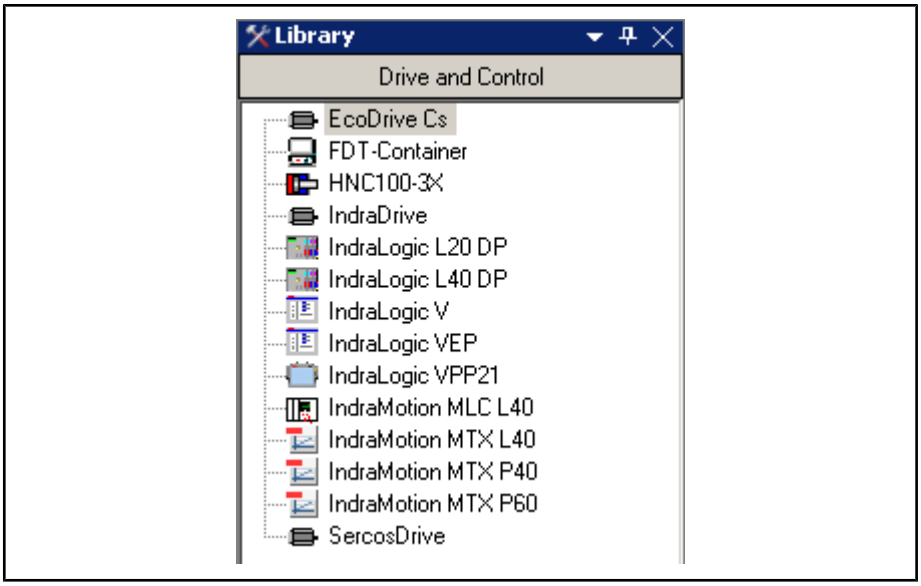


Fig.1-6: Devices in the library

1.3.6 Exiting IndraWorks

Exit IndraWorks via **File** ▸ **Exit** or by <Alt>+<F4>.

2 Working with IndraWorks

2.1 General Information

In IndraWorks, you will work with objects with various contents, called application data. These objects can be created, managed, edited and filed in a structured manner, as files in an IndraWorks project. The workspace in IndraWorks consists of a top-level project. Each project can, in turn, contain a collection of folders and files. These objects are also called resources.

2.2 Working with Projects and Devices

2.2.1 Projects and Project Data

In IndraWorks, devices and objects (resources) are compiled to projects which are represented in a tree structure in the project explorer. A project is always at the uppermost hierarchy level. Only one project can be processed at a time.

All further nodes in the tree represent devices, communication connections, functions or merely structuring elements, e. g. folders, which are comparable with directories in a file system. The figure below is an example of a project with a drive.

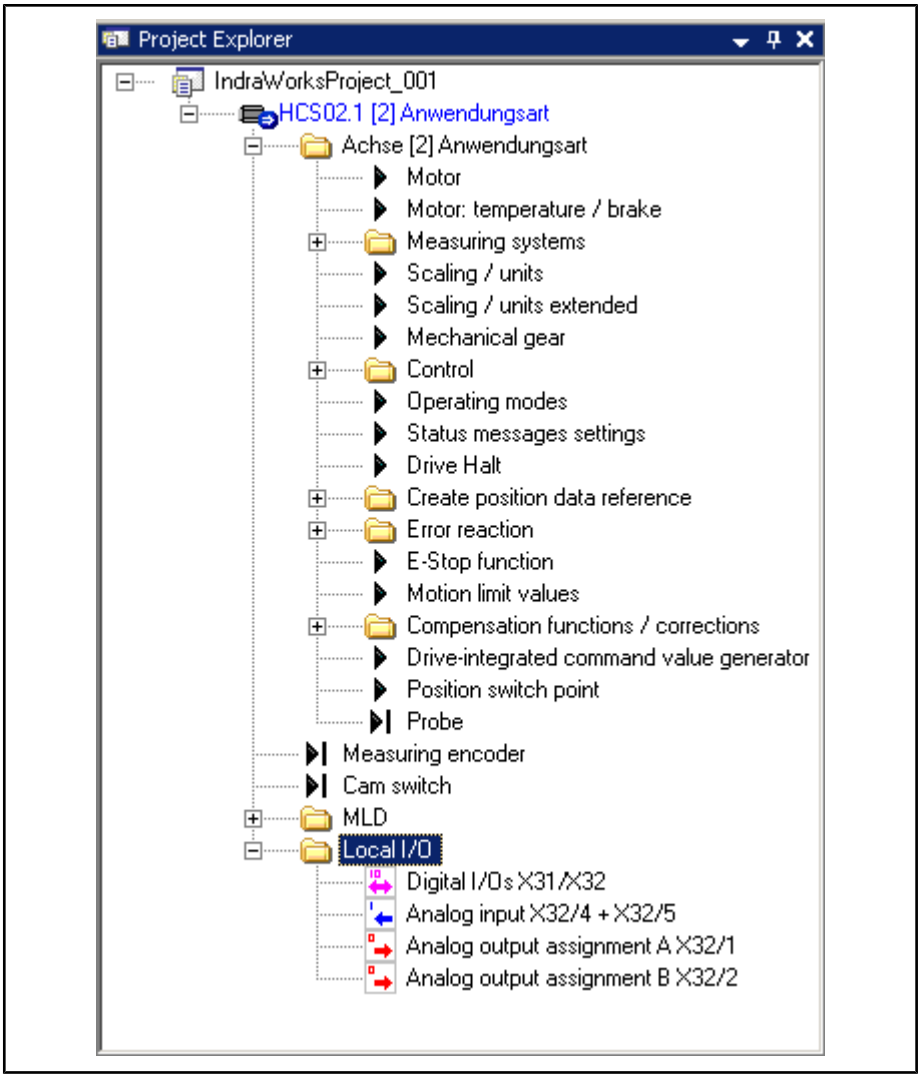


Fig.2-1: Project in a tree structure

Each node can provide a corresponding main menu entry with subitems and a context-sensitive menu. The content and function of the commands provided in this menu depend on the particular node.

If necessary, the “Project” toolbar can be shown. When shown for the first time, this toolbar is floating, but it can be moved to the desired position using the mouse.

Fig.2-2: Project toolbar

This toolbar provides the following commands:

- Switch project offline (see main menu **Project ▸ Switch Devices Offline...**)
- Switch project offline (see main menu **Project ▸ Switch Devices Online...**)
- Start offline parameterizations (see main menu **Project ▸ Start Offline Parameterizations…**)
- Scan for devices (see main menu **Project ▸ Scan for Devices...**)
- Archive project (see main menu **Project ▸ Archive**)

- Restore project (see main menu **Project ▸ Restore**)

Creating a New Project

To create a new project, select **File ▸ New ▸ Project...** or press <Ctrl>+<Shift>+<N>.

Enter a project name in the “Create New IndraWorks Project” dialog box, and select the directory where you wish to save the project. In the dialog box, select the master language for the project, i. e. the language used for creating the project.

You can also define the fonts for the text display, both for proportional fonts and for monospace fonts. This is particularly recommended for languages requiring specific fonts for displaying the text, e. g. Asiatic languages.

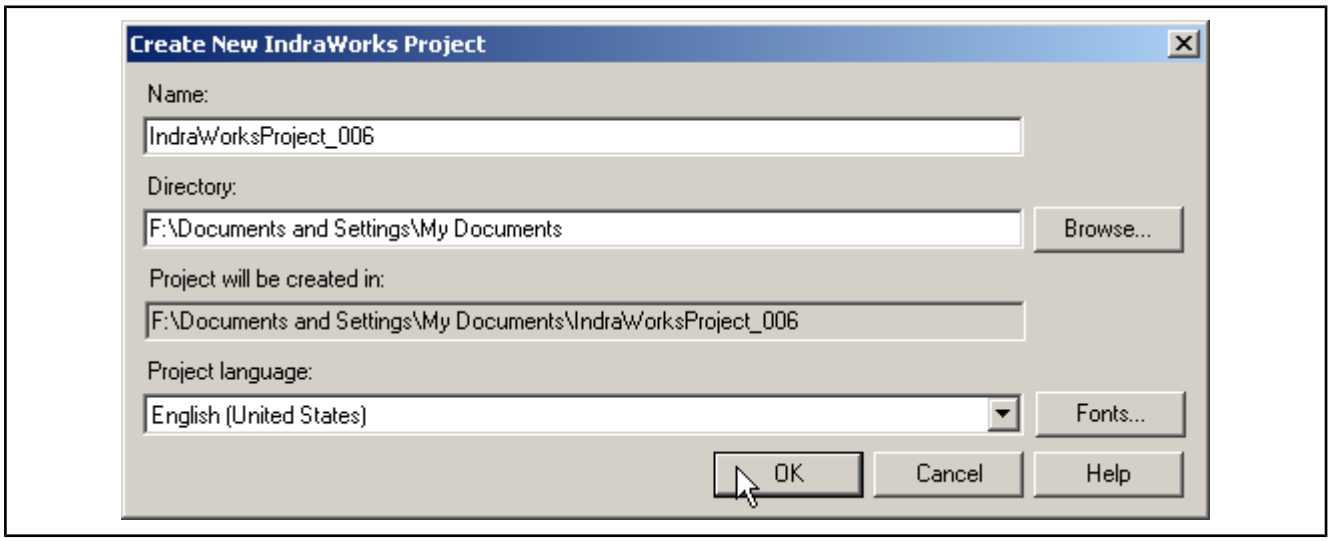


Fig.2-3: Create New IndraWorks Project dialog box, define master language

Opening an Existing Project

You can load existing projects via **File ▸ Open ▸ Project...** or <Ctrl>+<Shift>+<O>. The “Open Project” dialog shows all projects of the preset project path.

First select the folder of the project desired and then the project file. The file extension of project files is always “iwp”. The project is now uploaded.

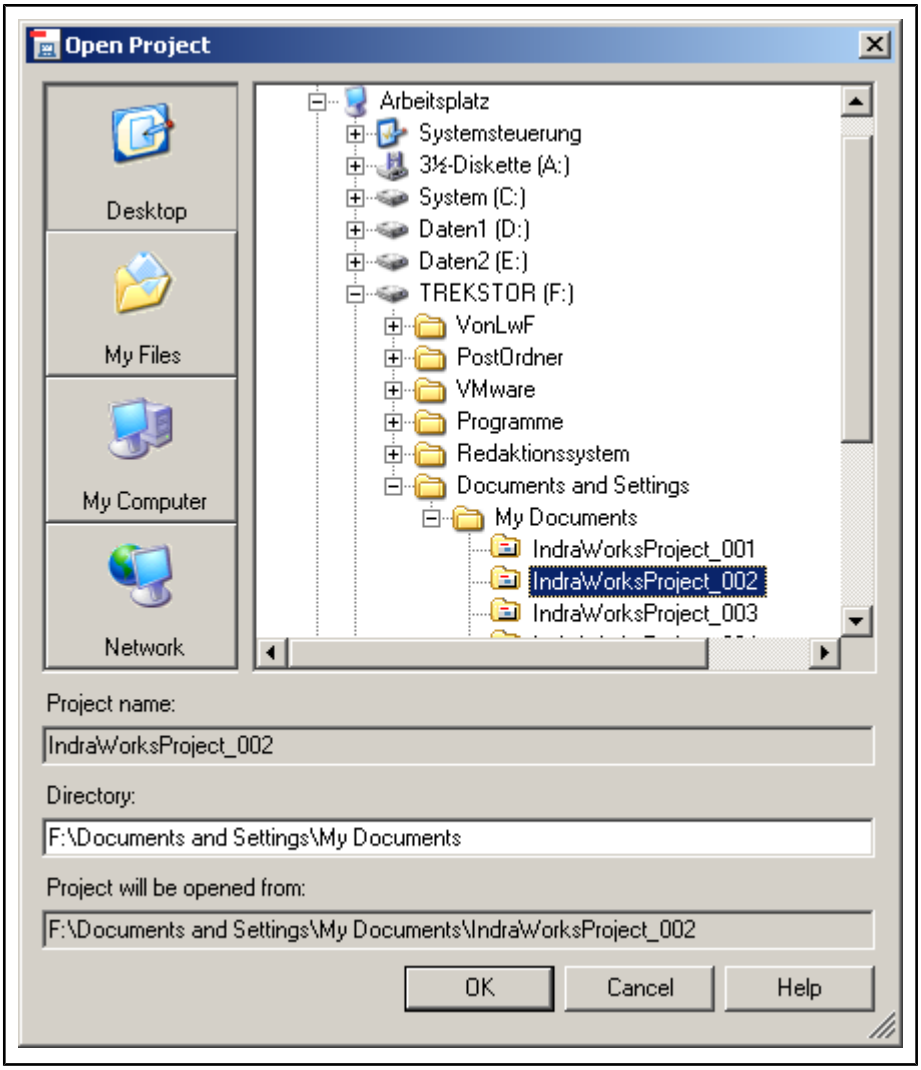


Fig.2-4: Open Project dialog box

Recent Projects

The menu item **File ▸ Recent Projects** provides a list of the projects you have processed recently. You can open a project by double-clicking on the respective entry in this list.

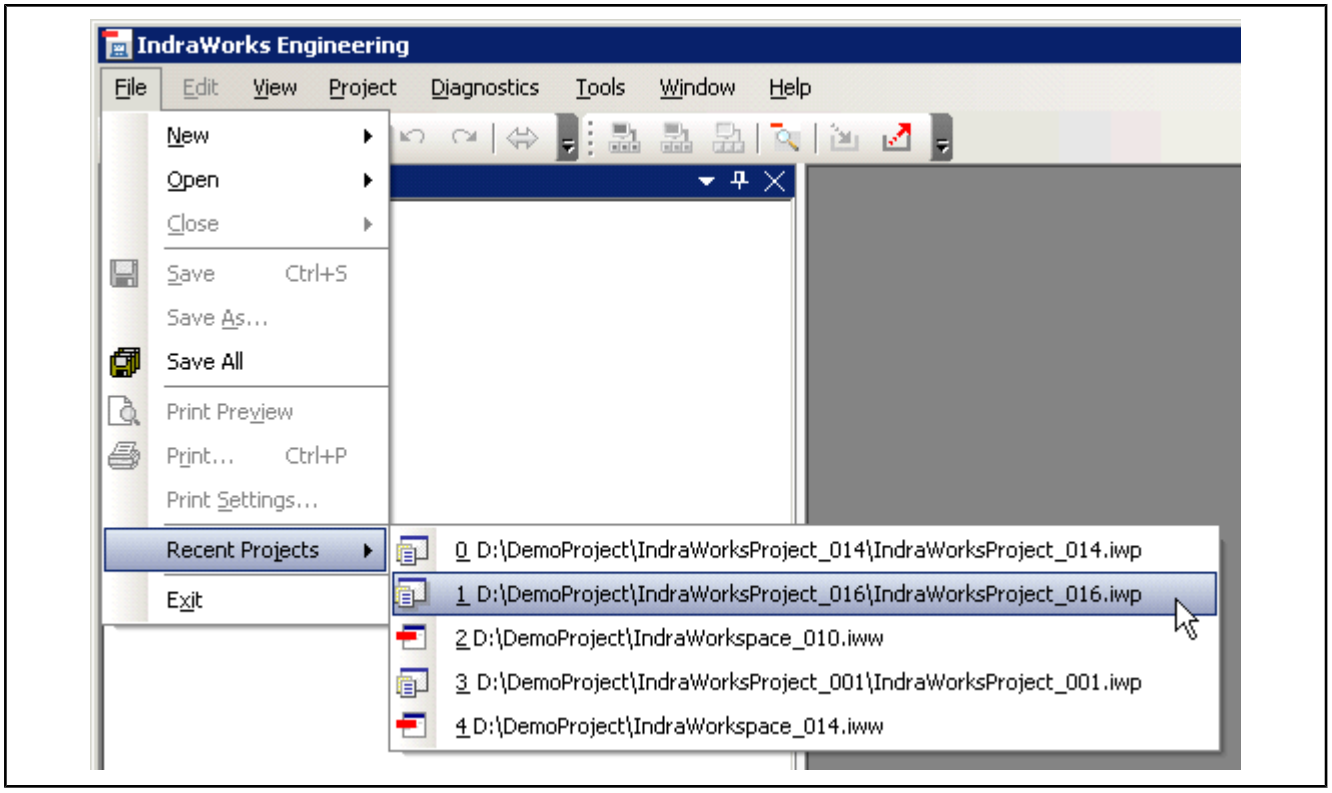


Fig.2-5: Recent Projects menu item

Closing a Project

Close an open project via **File** ▸ **Close** ▸ **Project**. All changes will be saved automatically.

Renaming a Project

You can change the project name by clicking on the activated project, by using the context-sensitive menu or by pressing <F2>.

Only the name of the project file “*.iwp” will be modified. The name of the path or directory where the project name is stored remains unchanged.

Properties

To open the properties dialog, select a project and choose the item **Properties** in the context-sensitive menu. The properties dialog box displays the essential settings of the current project. In the selection window to the left, you can move back and forth between path settings, modification times, states and version control.

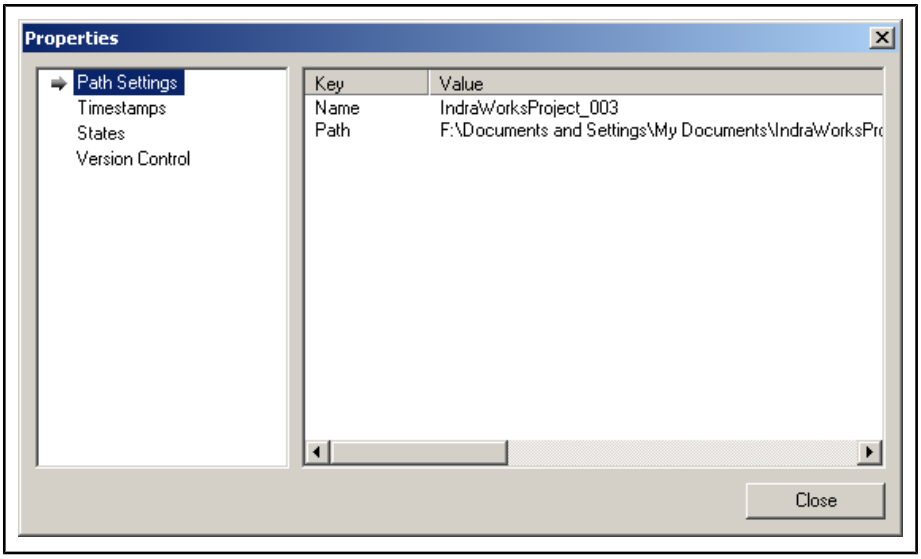


Fig.2-6: Properties dialog box, Path Settings

Selecting “Path Settings” will provide information on the name and the memory location of the project.

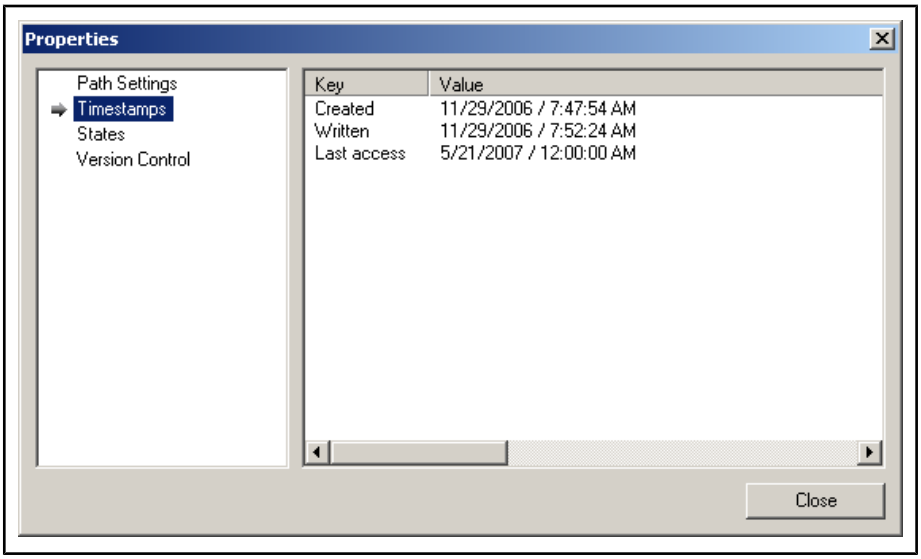


Fig.2-7: Properties dialog box, Times

Selecting “Times” will display the creation time and the time of the last write access and of the last read access.

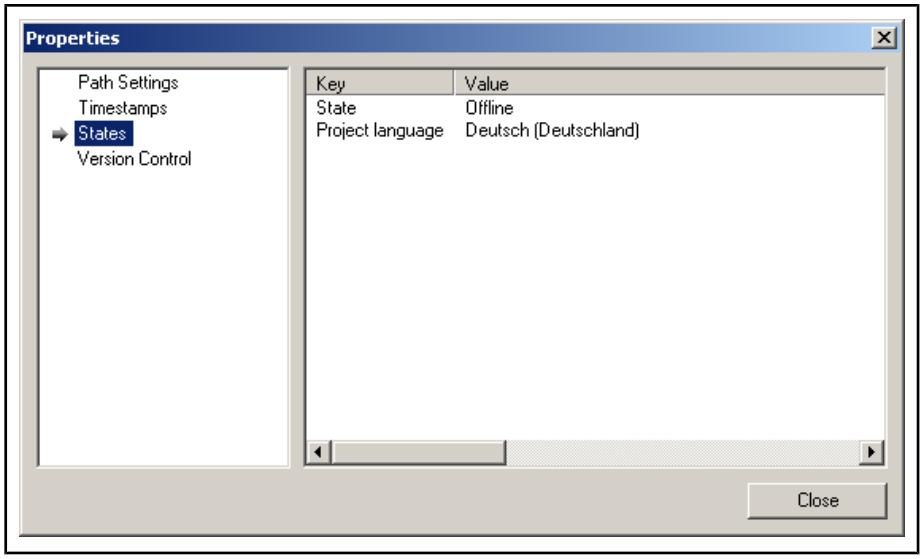


Fig.2-8: Properties dialog box, States

Selecting “States” will display the current state (offline, online) and the project language selected.

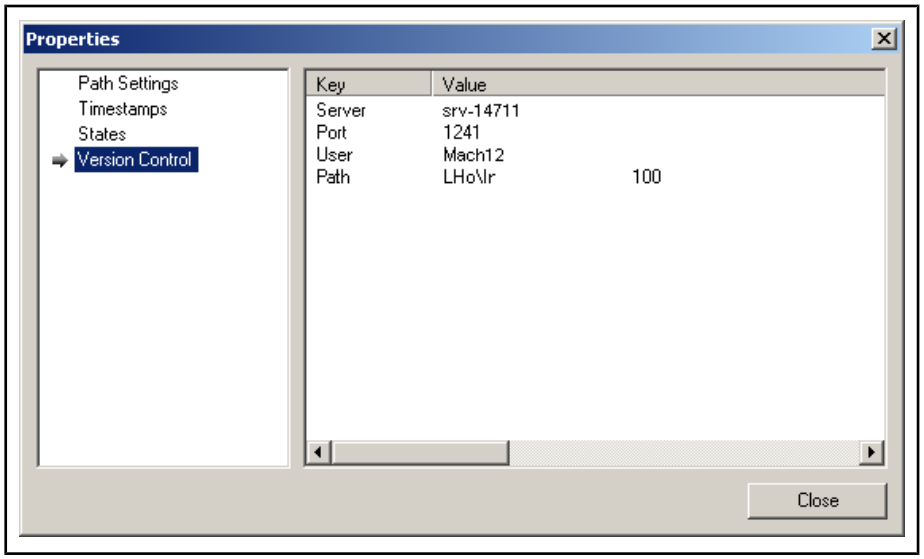


Fig.2-9: Properties dialog box, version control

Selecting “Version Control” will display information on the project management in the version control.

Save the project

An edited project is identified by an asterisk following the project name in the title bar of the project explorer. To save an edited or new project, select the item **Save** in the context-sensitive menu. When you close a project, all changes are saved automatically.

Saving a Project As

To save the open project to a different memory location with a different name, select **File ▸ Save As**.

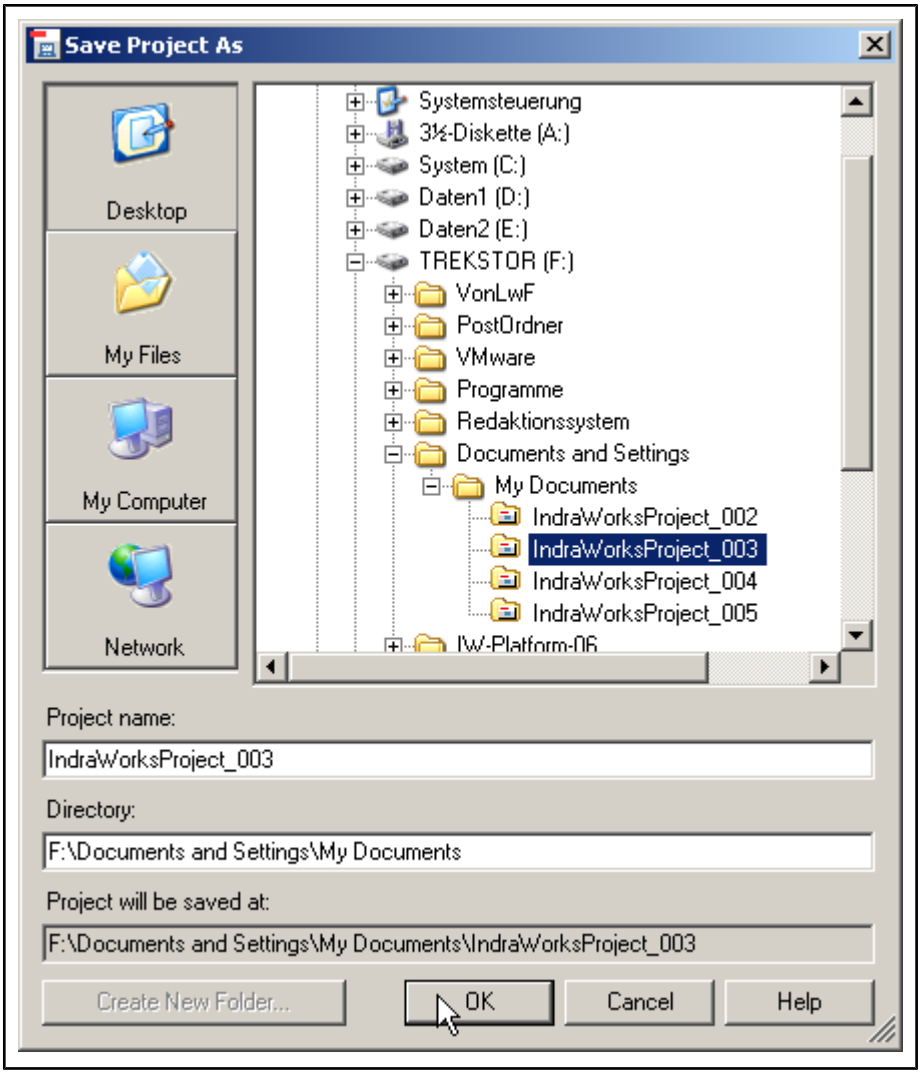


Fig.2-10: Save Project As dialog box

Enter a destination directory and a name for the project. After the dialog has been exited with “OK”, IndraWorks saves and closes the current project and creates a copy with the selected name in the specified destination directory.

☞ All changes in the project structure and in the project data made up to that point will also be saved in the original project.

The project information file “*.iwp” is not stored in the specified destination directory, but in a new directory with the same name.

If the selected destination directory already exists, the process will be stopped and a corresponding message is displayed

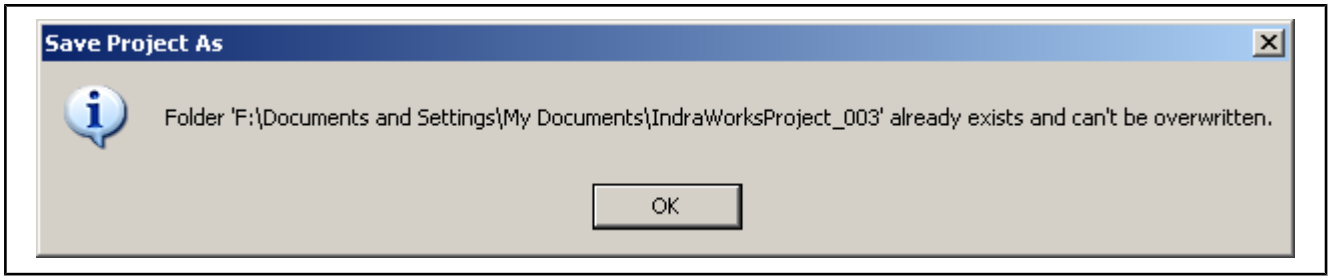


Fig.2-11: Save Project As, error message

2.2.2 Startup Picture

Unless a project has been uploaded, the startup picture will be displayed automatically when the Engineering Desktop is started.

This occurs in the follwing cases:

- IndraWorks has been installed anew.
- The project was closed before IndraWorks was exited the last time.
- The setting causing the previous project to be uploaded on start has not been activated.

The startup picture provides various options of creating or opening a project. To activate an option, click on it or select it with <Tab> and <Enter>.

Once a project is uploaded, the startup picture is closed. You can show the startup picture at any time via **View ▸ Show Startup Picture**.

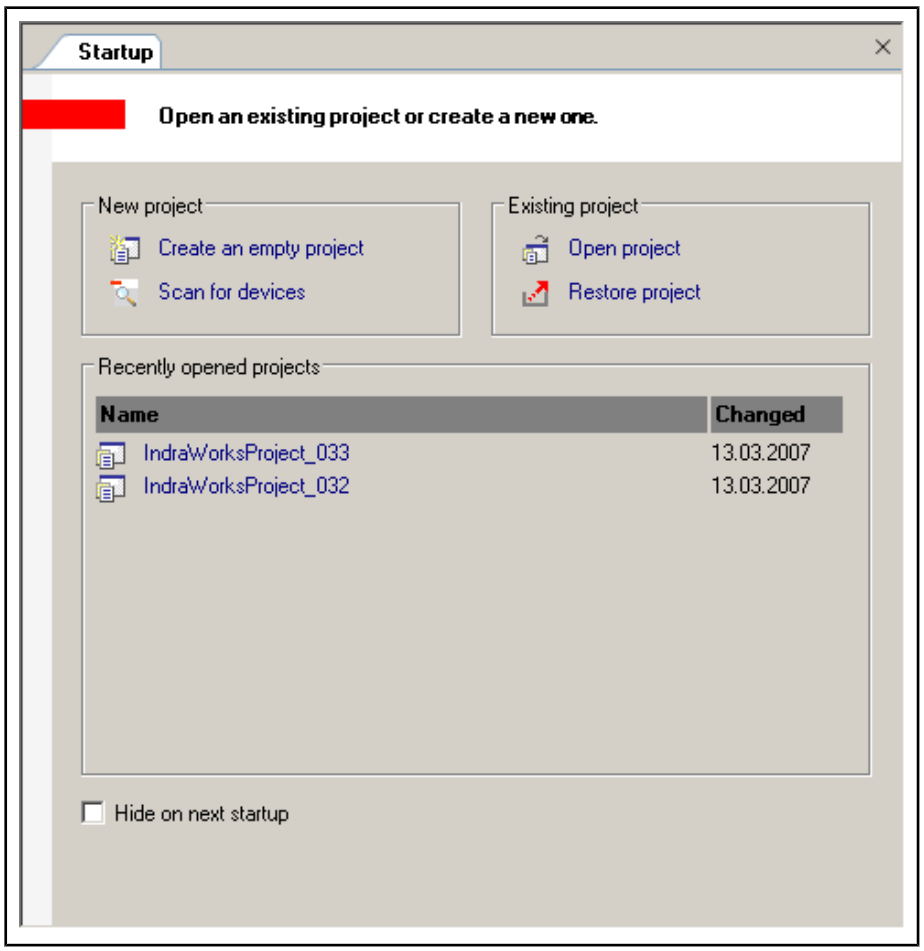


Fig.2-12: Startup Picture

Create an Empty Project This function opens the “Create New IndraWorks Project” dialog box. It is equal to **File** ▸ **New** ▸ **Project**.

Scan for Devices This function creates an empty project with a defined name and starts the scanner to search for devices. Simultaneously, the view “Configure” will be activated (see chapter 2.2.8 "Scan for Devices" on page 40).

Open Project This function opens a project which is filed to a local drive or a network drive. It is equal to **File** ▸ **Open** ▸ **Project**.

Restore Project This function restores a project from an archive on a local drive, a network drive or a removable disk. It is equal to **Project** ▸ **Restore** ▸ **From File System**.

Recent Projects The list of recently opened projects displays the project names and the changing date. To open a project, click on the project name. This function is equal to **File** ▸ **Recent Projects**.

Hide on Next Startup If you activate this option, the startup picture will not be displayed automatically when the Engineering Desktop is started.

2.2.3 Devices

Inserting Devices From the Library

Add devices to a project by drag-and-drop or via the context-sensitive menu.

context-sensitive menu The context-sensitive menu provides the commands available in the particular context. The commands for editing devices, such as **Cut**, **Delete**, **Copy** and **Paste**, are also provided in the **Edit** menu. In contrast to the context-sensitive menu, inactive items are visible in the main menu. They are displayed in gray.

Insert by Drag-and-Drop from the Library Select a device from the library and drag it to the project explorer.

The form of the mouse pointer indicates possible insertion positions.

An arrow with a plus sign on a destination device with blue background (including the project itself) signals that the device can be inserted at this position.

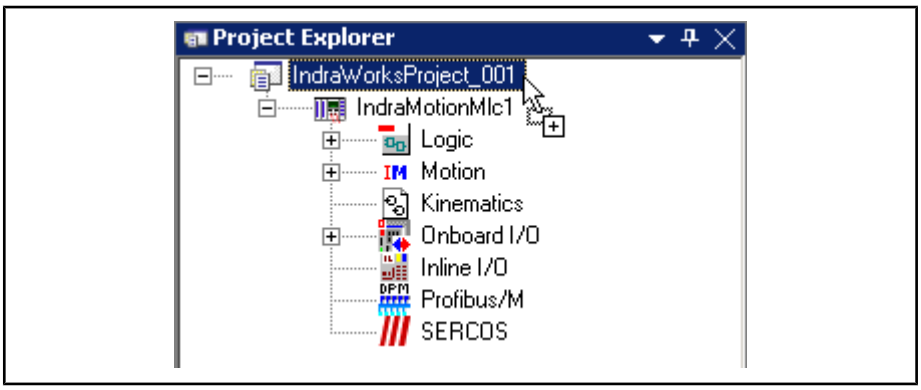


Fig.2-13: Insert by Drag-and-Drop from the Library

The new device is added behind all devices of this element.

A circle with a backslash signals that it is not possible to insert a device at this position.

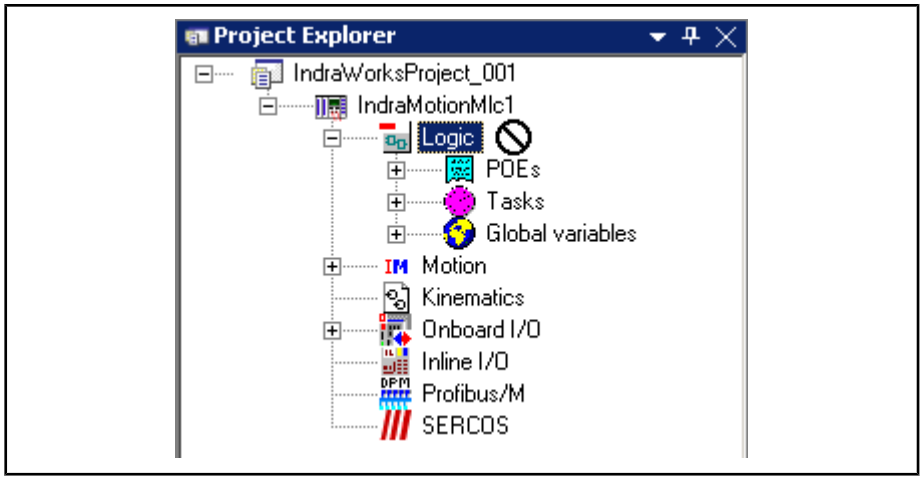


Fig.2-14: Insertion by drag-and-drop not allowed

You can also insert new devices in the project at certain selected positions.

Move the mouse to the desired destination position. This position is represented by a line. The background color of the associated destination element changes to blue.

In the first example, the new device is inserted below the SERCOS node.

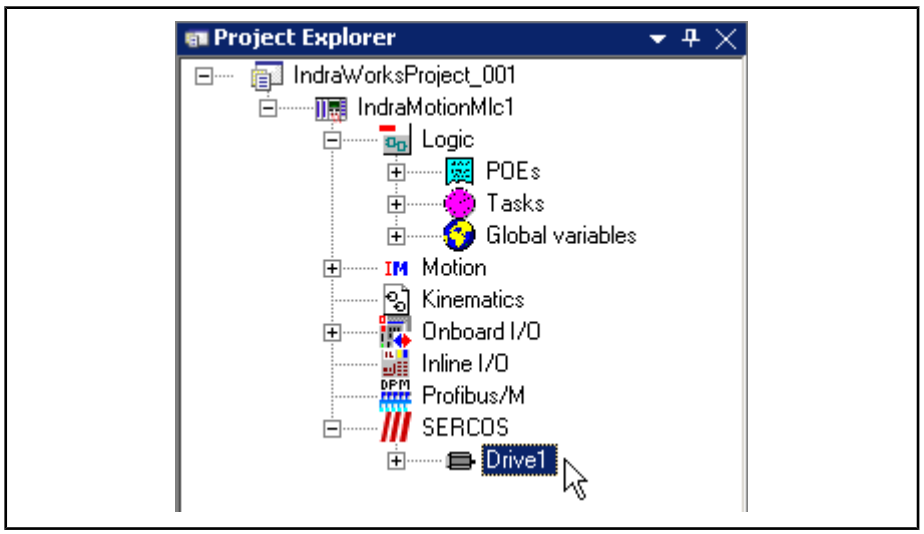


Fig.2-15: Insertion at the desired destination position

In the second example, the new device is inserted below the project node cret-aed before.

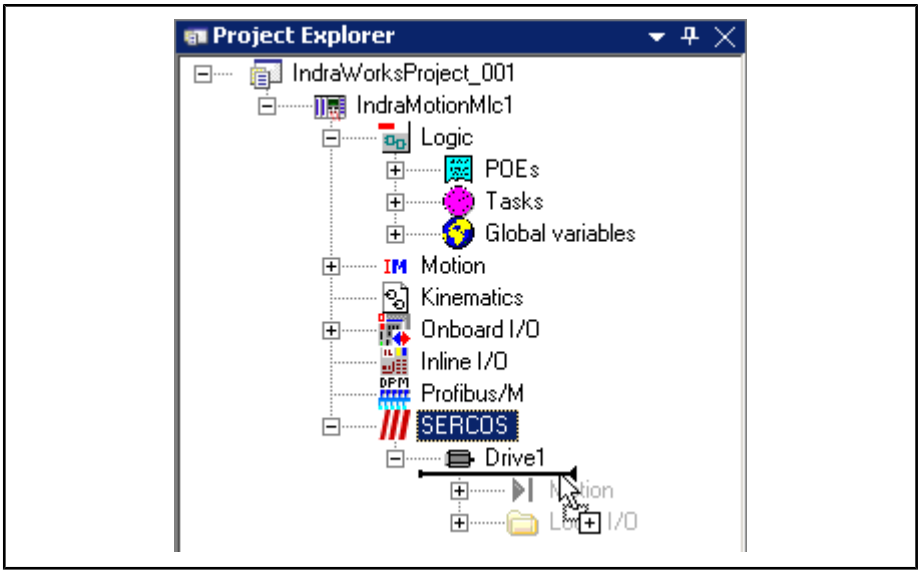


Fig.2-16: Insertion at the project node

Insert via Clipboard

You can also use the clipboard to add devices to a project.

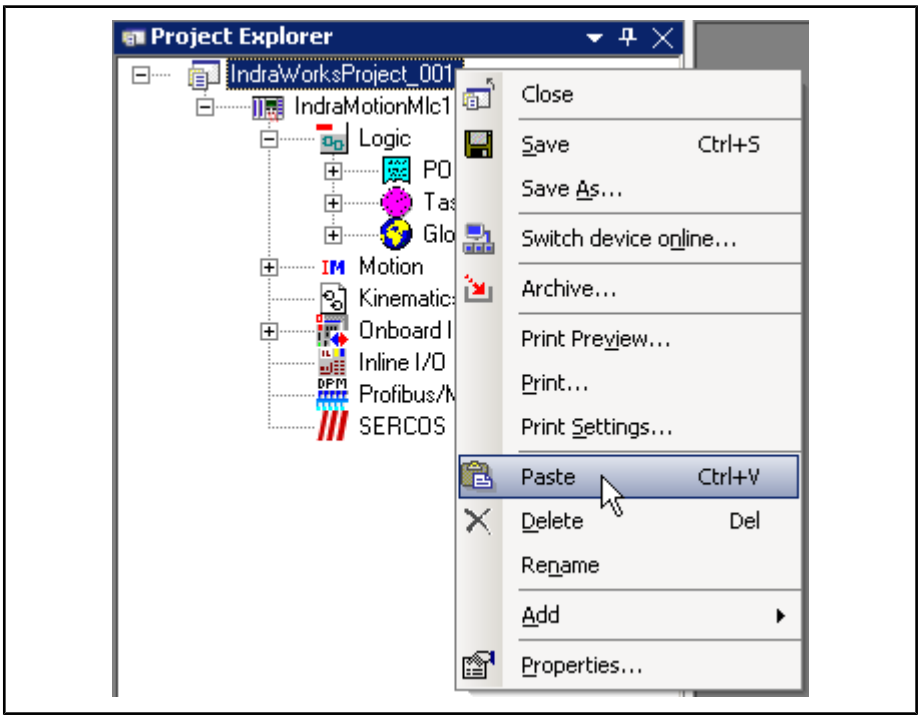


Fig.2-17: Pasting devices from the clipboard

You can also paste devices from foreign components to the project explorer via the clipboard. This requires a complete description of the device as XML text in the clipboard .

Devices in the Project Explorer

Deleting Devices

Delete devices by <Del> or use **Delete**.

Cutting Devices

Cut files a reference to the selected device to the clipboard.

The device prepared for cutting is identified by a special icon (arrow to the upper right) and by a gray font as long as the device information is in the clipboard. You can stop this process with <Esc>.

If you paste the device at a new position, it will be removed from its original position.

Copying Devices

Copy applies a copy of the selected element to the clipboard. You can now paste the element to the destination position.

It is possible to paste the device directly to an element or to positions between elements.

Copying, Cutting and Pasting with the Mouse

You can copy, cut and paste devices by drag-and-drop. Drag a device to another possible position and it will be moved to that position. If you hold <Ctrl> while dragging, a copy of the device is created at the destination position.

Renaming Devices

You can change the device name by clicking on the activated element, by the context-sensitive menu or by <F2>.

The name of the element is displayed in the input mode. You can stop the editing process at any time with <Esc>. If the entered name is not accepted, the original name is automatically re-entered.

Opening Node Specific Dialogs and Editors

To open a dialog or editor associated to a project node, double-click on that project node or press <Enter>.

Tool Tips

To show brief information on a device, place the mouse pointer on that device for more than one second.

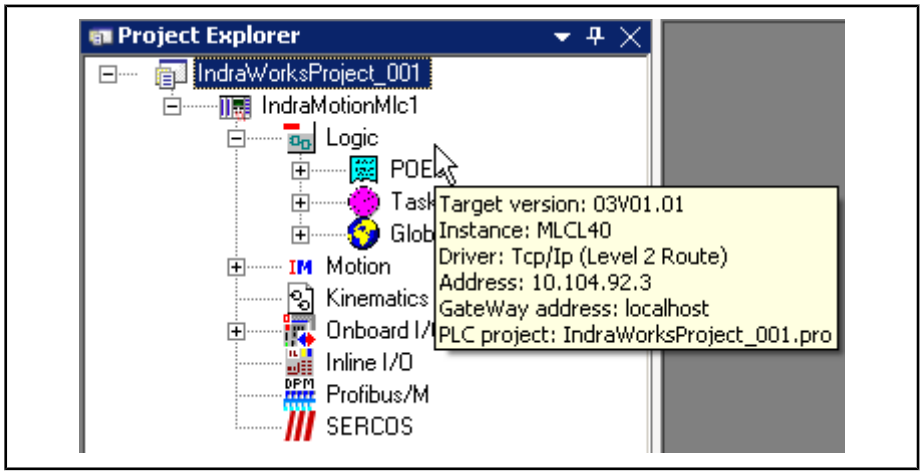


Fig.2-18: Displaying tool tips for devices

Displaying Error States

If the device-specific software signals an error, the device in question is shown with red letters. If this device is in a non-expanded part of the tree, the device that is visible at the next higher level in the hierarchy is shown in red. The incorrect element itself is identified by a white cross on a red background in the icon. Additionally, an error text is provided.

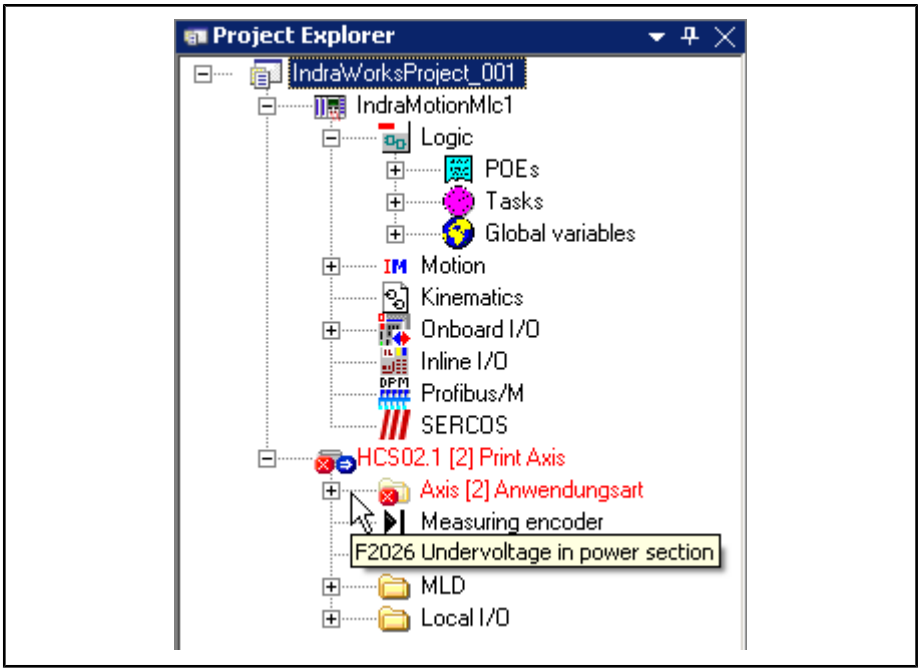


Fig.2-19: Displaying error states at devices

This shows incorrect elements immediately in the tree structure.

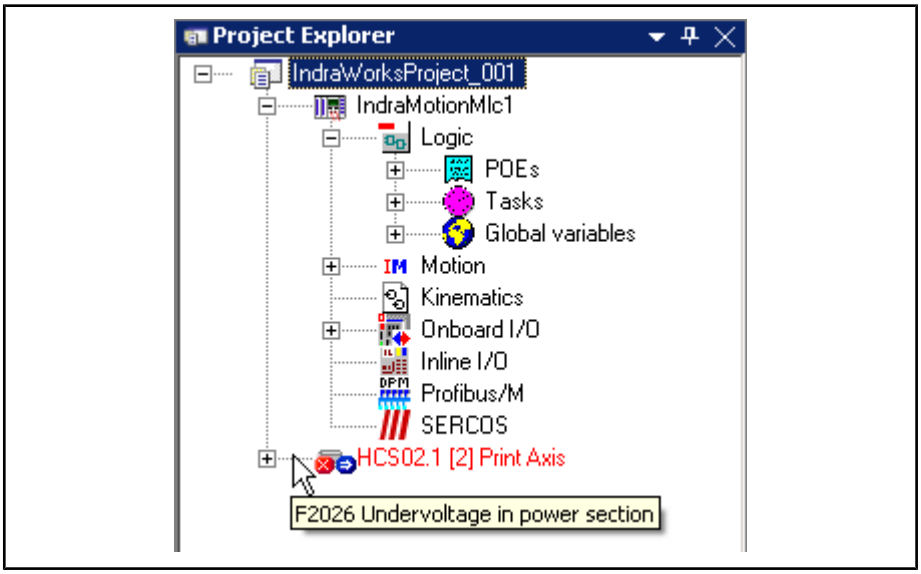


Fig.2-20: Displaying error states at device nodes

Locked Elements

An element can be locked if, e. g., it cannot be reached in the particular context. Locked elements are displayed in light-gray in the tree structure; the icon of the device is lightened. Such elements cannot be reached or modified by commands any longer.

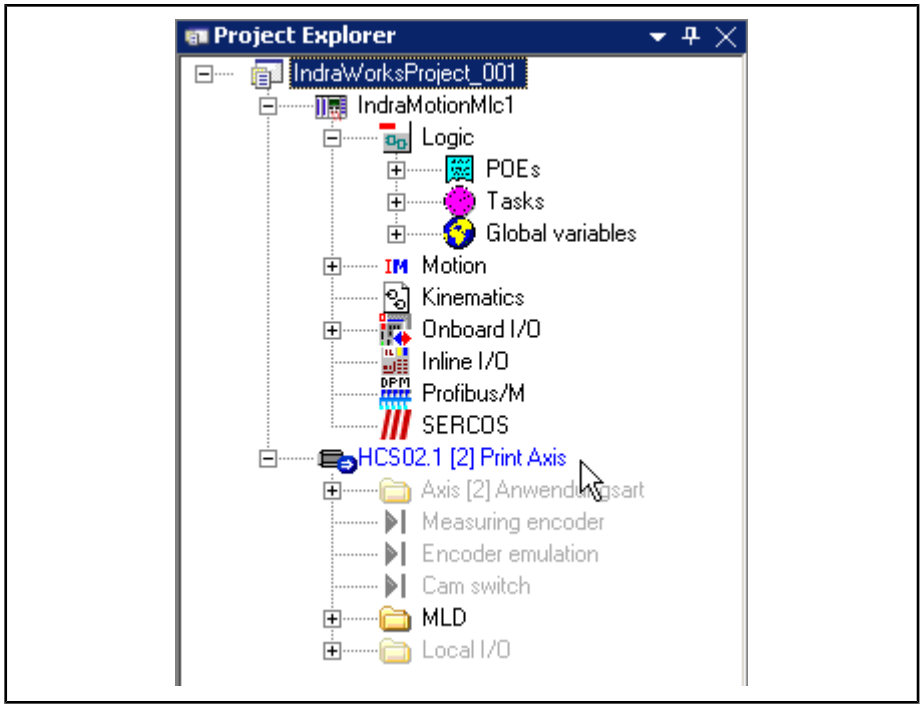


Fig.2-21: Locked elements in the project explorer

Restructuring with Folders

Use the context-sensitive menu **Add ▸ New Folder** to add new folders to the structure at possible positions.

Folders can be moved, copied, deleted and renamed. Additionally, you can open parallel partial views of the project via the context-sensitive menu of the folder.

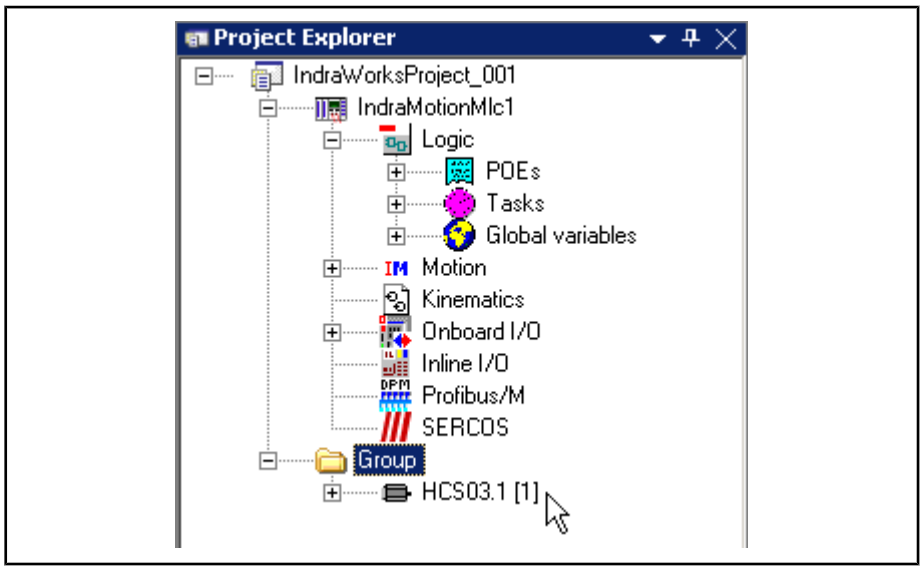


Fig.2-22: Sructuring with folders

2.2.4 Using the Library

General Information

The library contains all devices available in the platform. The library uses data from the standardized device description file, which is required for configuration and parameterization. This data is filed to the “Runtime\Library\Devices\Group name” directory in a structure and subdivided according to manufacturers.

Function Areas

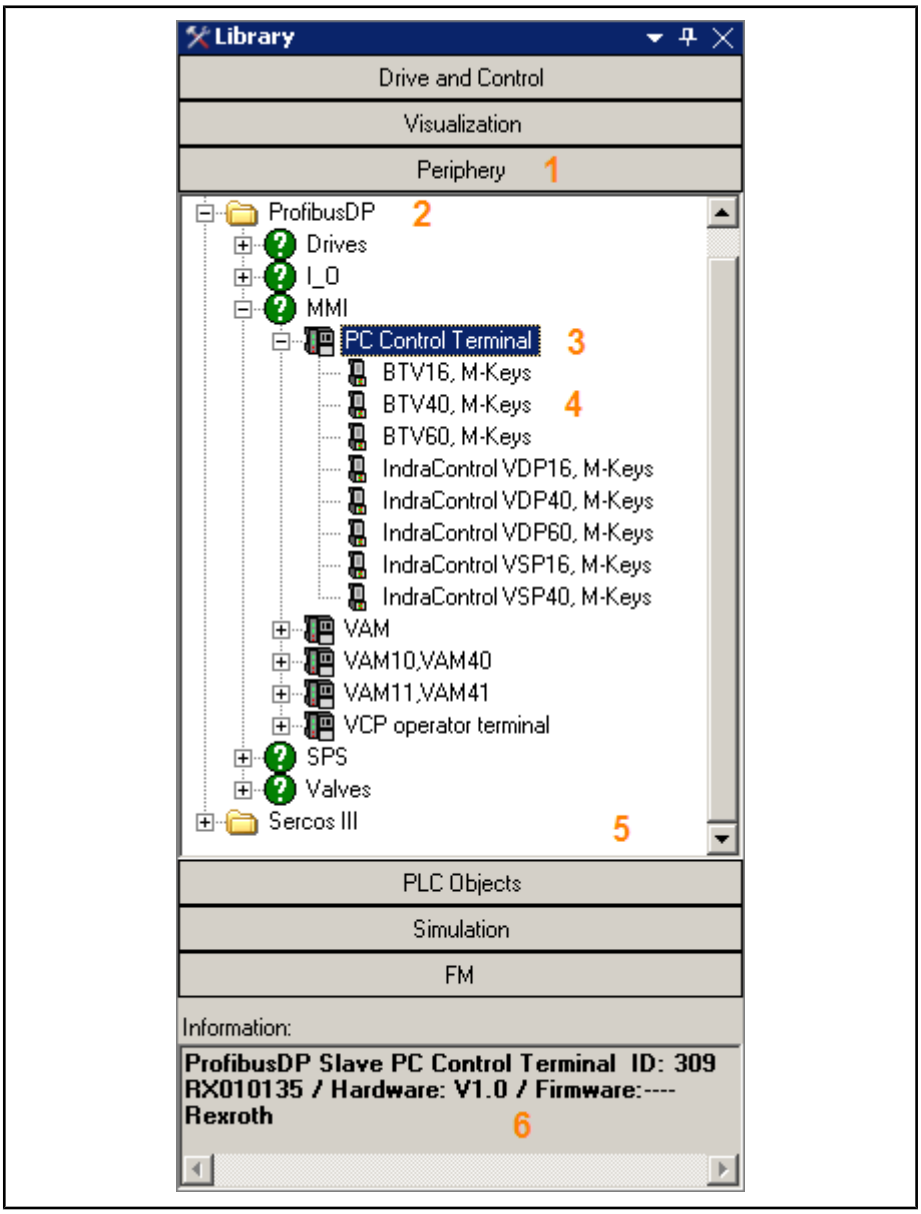
The library is divided into a navigation area with the tree structure of all installed devices and into an area providing current information on the selected device.

Navigation Area

The navigation area is divided in library groups. The various groups are subdivided according to manufacturers and functional components. The devices are defined by their name and the device-specific icon.

Information Area

If you have selected an element in the navigation area, current data on the device is displayed in the information area.



1 Library group with its own tree
2 Folders
3 Devices
4 Module for the device
5 Navigation Area
6 Information Area

Fig.2-23: Function areas of the library

Operation

Navigation

You can navigate in the tree structure of the devices with the cursor keys and with the mouse.

To expand nodes at the “plus” sign, press <cursor right> or double-click; to close the nodes at the “minus” sign, press <cursor left> or double-click.

Adding Devices to the Project

Use drag-and-drop to add devices from the library to the project navigator. (see chapter "Inserting Devices From the Library " on page 18) After insertion, the software package of the device is started.

You can also use the clipboard by **Copy** and **Paste** to add devices.

Expanding the Library

Devices are added by copying new device description files or complete structures to the “Runtime\Library\Devices\Group name” directory. These data will be automatically applied to the graphical display when the platform is started the next time. Copying data on the level of library groups is not allowed.

2.2.5 Archiving and Restoring Projects

General Information

IndraWorks provides the option of archiving projects on the local file system or on an FTP server (device or computer) connected through a network. These archives can be restored on the file system of the local computer.

A wizard supports the working with project archives. If the entered values are correct, you can move between the pages of the wizard by “<<Back” and “Next>>”. When a page is opened for the first time, the input boxes contain default values. Otherwise, your last entries will be displayed. You can exit the wizard at any time with “Cancel”. Values entered up to that point will not be saved; the archiving process is stopped.

Archiving Projects

To archive a project, select it in the project explorer and choose **Archive…** from the context-sensitive menu, **Project ▸ Archive** in the main menu, or the following button from the toolbar:

Fig.2-24: Project toolbar

Archiving a Project on a File System

Selecting the Archiving Type

The first page of the wizard prompts you to select whether you wish to save the archive on a local file system or on an FTP server (device or computer) connected through a network. Select here “Archive on file system”.

Settings for Destination Archive

On the next page you can define the filing location, the name of the archive and a comment. Enter a directory of the local file system for the filing location or select it via “…”.

Optionally, the archive can be protected by a password. Enter the password a second time in “Confirm password” to verify your entry.

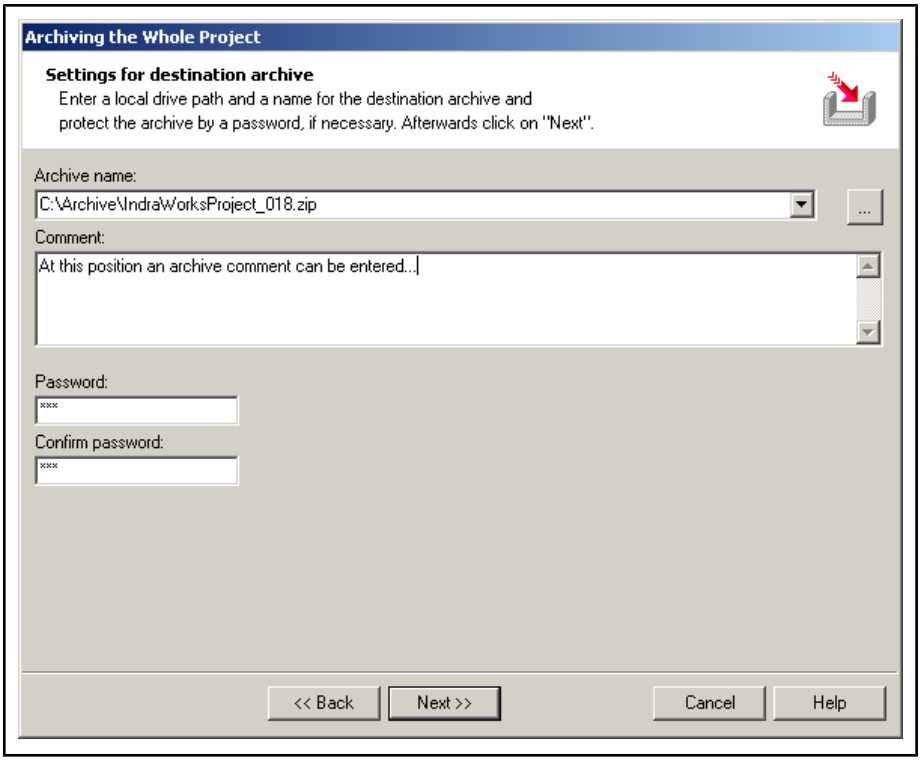


Fig.2-25: Archiving a project on a file system; destination archive settings

Checking the User Entries

This page allows to check your settings. Start the creation of the archive via the “Finish” button.

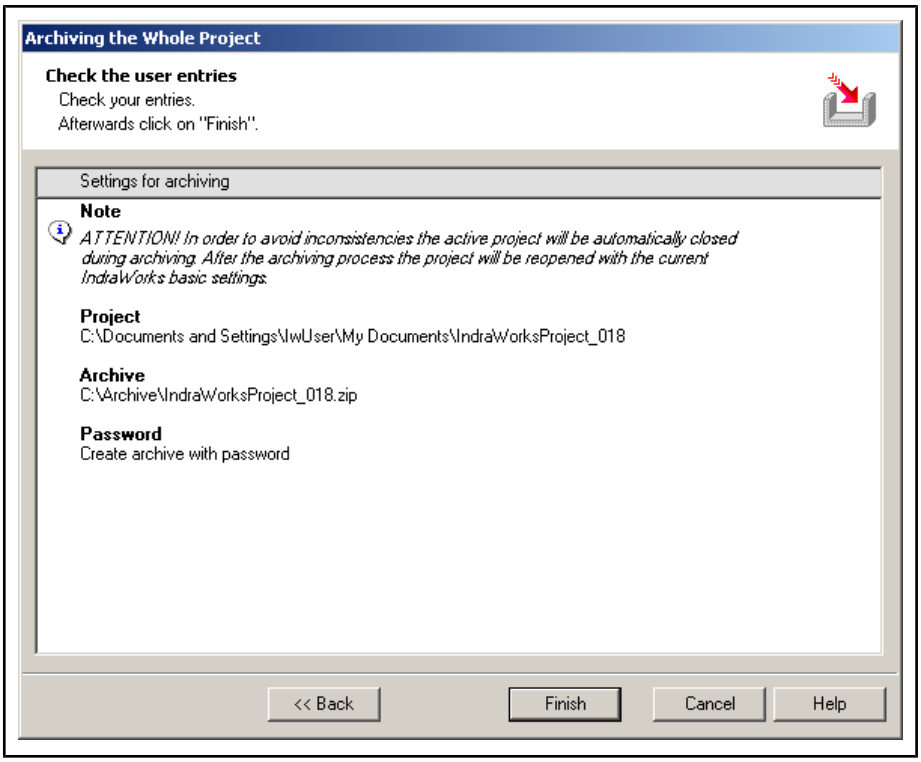


Fig.2-26: Archiving a project on a file system; check of entries

Progress Bar “Create Archive”

The archive is created in the destination directory of the local computer. This process is displayed in a progress bar.

Summary After the archiving the settings and results are displayed.

To avoid inconsistencies during archiving, the active project is closed before archiving and re-opened afterwards.

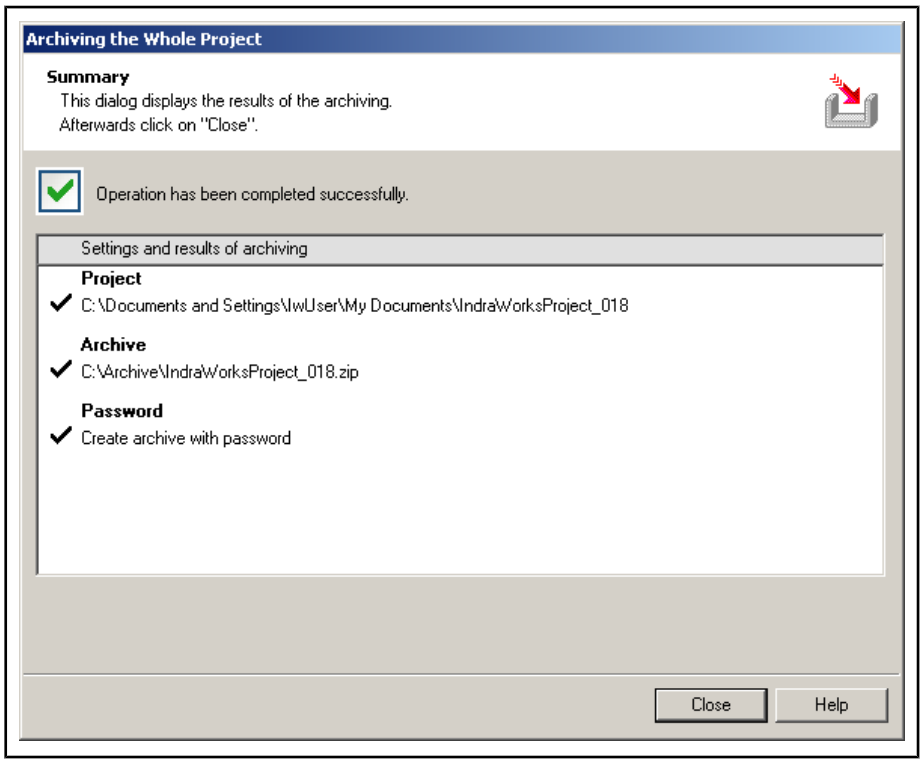


Fig.2-27: Archiving a project on a file system; archiving results

Archiving a Project on a Device

Selecting the Archiving Type The first page of the wizard prompts you to select whether you wish to save the archive on a local file system or on an FTP server (device or computer) connected through a network. Select here “Archiving on FTP server (device or computer)”.

Settings for Destination Archive On this page, you can define the destination device, the name of the archive and a comment.

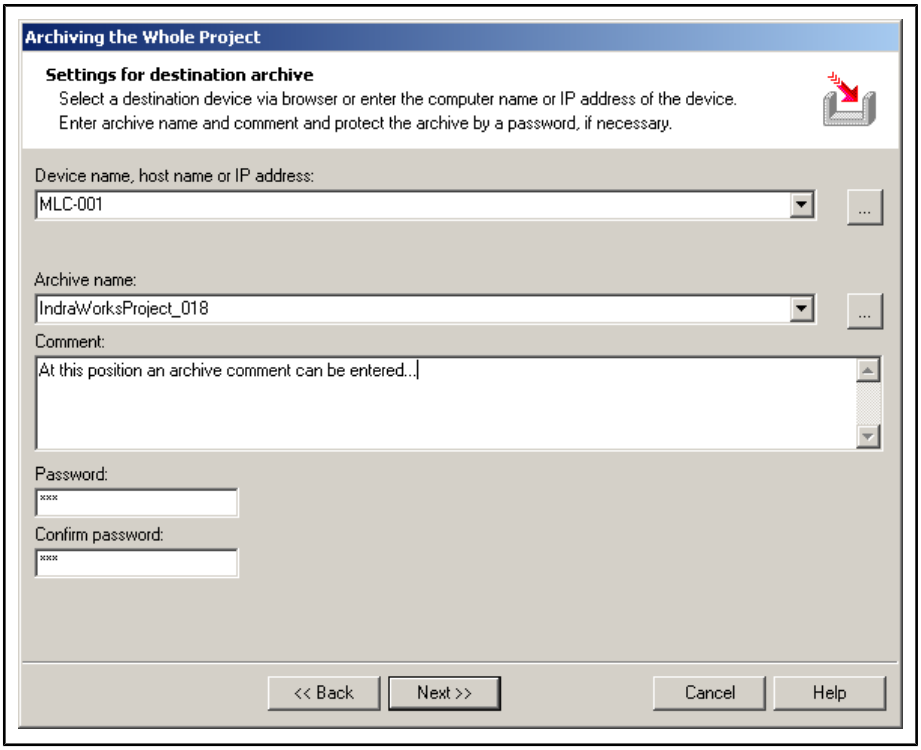


Fig.2-28: Archiving a project on a device; settings for destination archive

Device Name, Host Name or IP Address

Enter the destination device for saving the archive in the “Device name, host name or IP address” input box. You can do this in four ways:

- Enter the IP address (nnn.nnn.nnn.nnn)
- Enter the computer name of the destination device
- Select the destination device via a drop-down list. This list box contains all FTP capable devices of the active project as well as the five destination devices (device name, host name or IP address) last used in archiving.
- Select a device via the “…” browser button. Apply the destination device from the list of all FTP capable devices of the active project.

Archive Name, Comment, Password

Enter a name for filing the archive to the destination device. You can also enter a comment related to the archive.

Optionally, the archive can be protected by a password. Enter the password a second time in “Confirm password” to verify your entry.

Establishing the Connection

Confirm with “Next>>”. The wizard will automatically establish a connection to the destination device. Disturbances in the connection to the destination device are displayed in error messages.

Checking the User Entries

This page allows to check your settings. Start the creation of the archive via the “Finish” button.

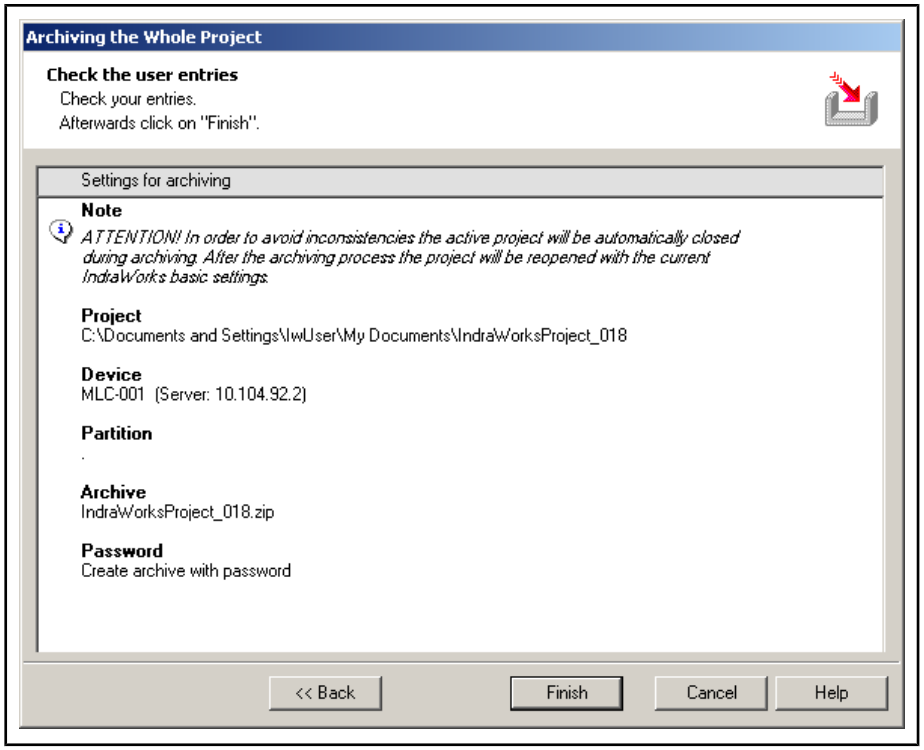


Fig.2-29: Arcive a project on a device; check of entries

1. Progress Bar "Create Temporary Archive"

On archiving, a temporary archive will be created on the local computer first. This process is shown in a progress bar.

2. Progress Bar "Copy Archive to Destination Device"

Then, the archive is copied to the destination device. This process is shown in another progress bar.

Summary

After archiving has been completed, settings and results are displayed in a summary.

To avoid inconsistencies during archiving, the active project is closed before archiving and re-opened afterwards.

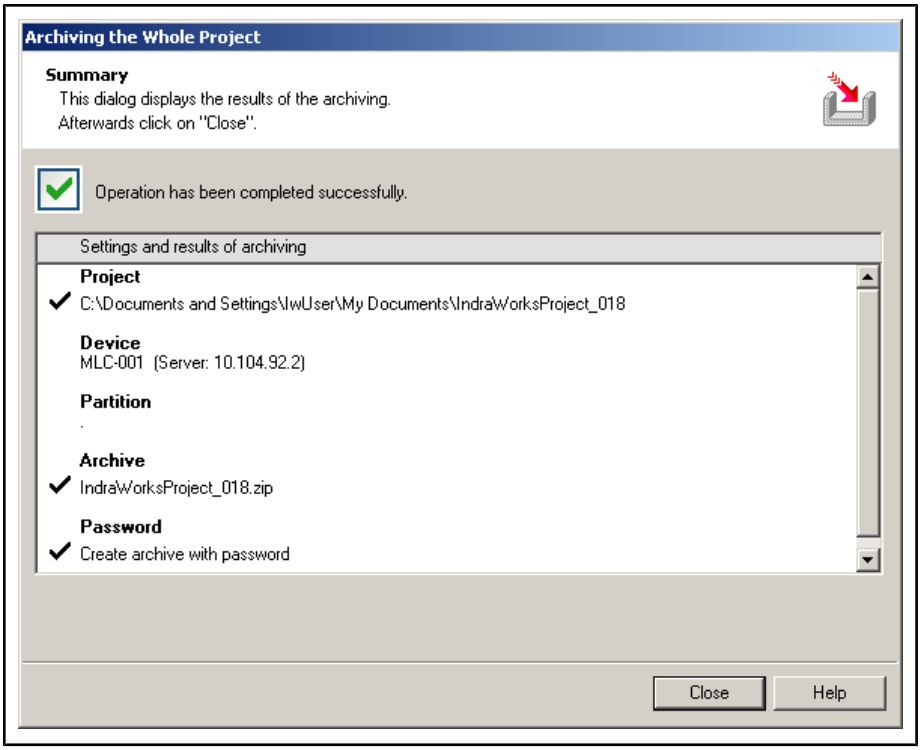


Fig.2-30: Archive a project on a device: results of archiving

Restoring Projects

To restore a project, select **Project** ▸ **Restore...** from the main menu or the following button in the toolbar:

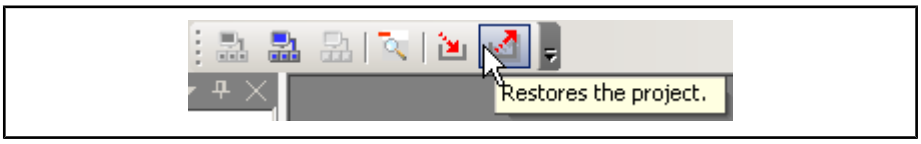


Fig.2-31: Project toolbar

Restoring a Project from a File System

Selecting the Restore Type

On the first page of the wizard, you can select whether to restore the project from an archive of the local file system or from an FTP server (device or computer) connected through a network. Select here “Restore from file system”.

Selecting the Archive

Select the archive on the next page. Click on the “…” button to search for the archive.

A comment will be displayed for to the archive selected.

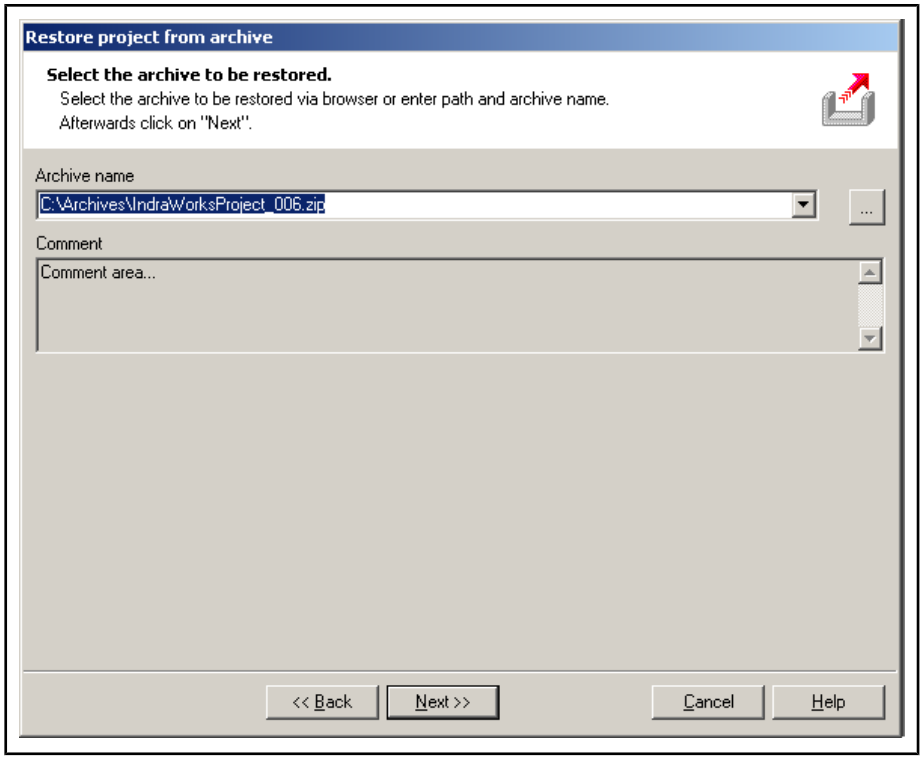


Fig.2-32: Restoring a project from a file system; selecting the archive

☞ If the archive type is unknown, the comment area will display the message “***ATTENTION! The selected archive is not an IndraWorks project archive ***”. In this case, you can continue the restore process after having confirmed a safety prompt.

Selecting the Destination Directory

On the next page, select the directory to which you wish to restore the project.

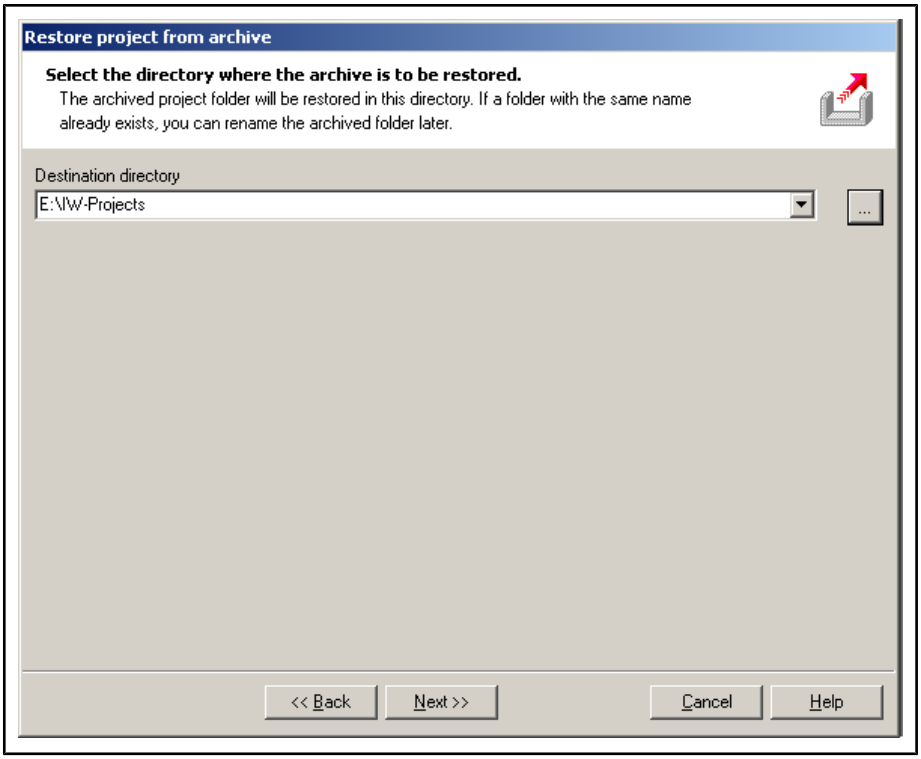


Fig.2-33: Restoring a project from a file system; select destination directory

Checking the User Entries

Here you can check your settings. Start the restore of the project from the archive by “Finish”.

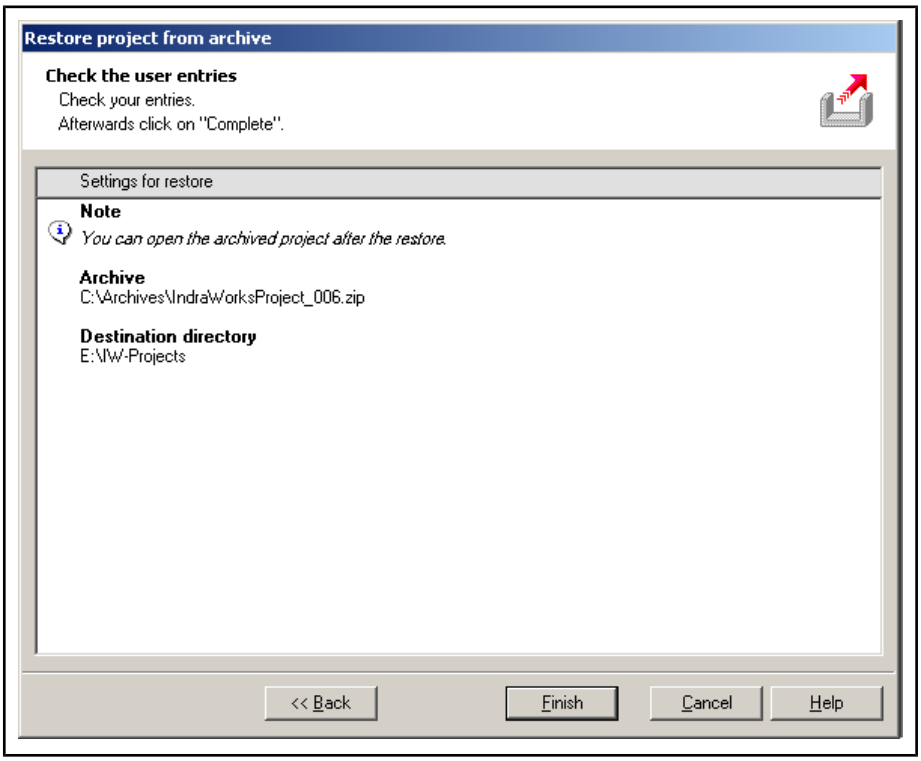


Fig.2-34: Restoring a project from a file system; check of entries

Entering the Password

If you have protected the archive with a password, you will now be prompted to enter that password.

Progress Bar “Restore on Temporary Directory”

First the project is restored from the archive to a temporary directory of the local drive. This process is displayed in a progress bar. After restore, the project is copied to the destination directory.

If a project folder already exists in the specified destination directory, you will be prompted to rename the project folder.

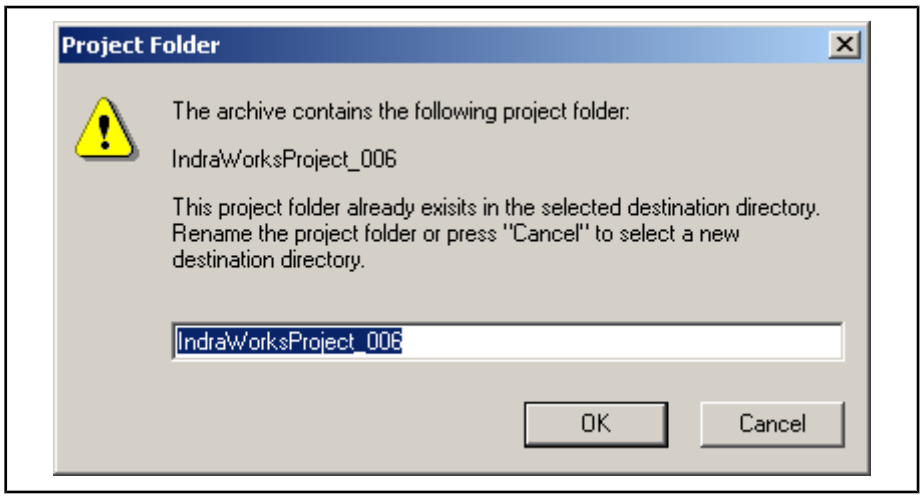


Fig.2-35: Restoring a project from a file system; renaming the project folder

Summary

After restoration, settings and results are displayed.

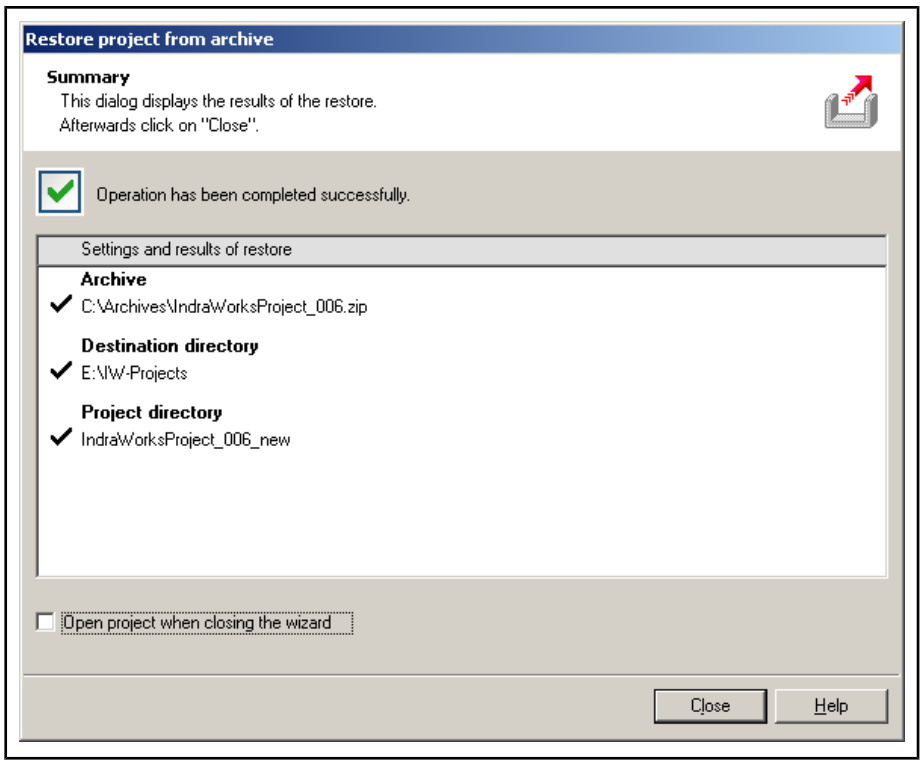


Fig.2-36: Restoring a project from a file system; results

Restoring a Project from a Device

Selecting the Restore Type

On the first page of the wizard, you can select whether to restore the project from an archive of the local file system or from an FTP server (device or computer) connected through a network. Select here “Restore from FTP server (device or computer)”.

Selecting the Archive

Select the device and the archive name on this page.

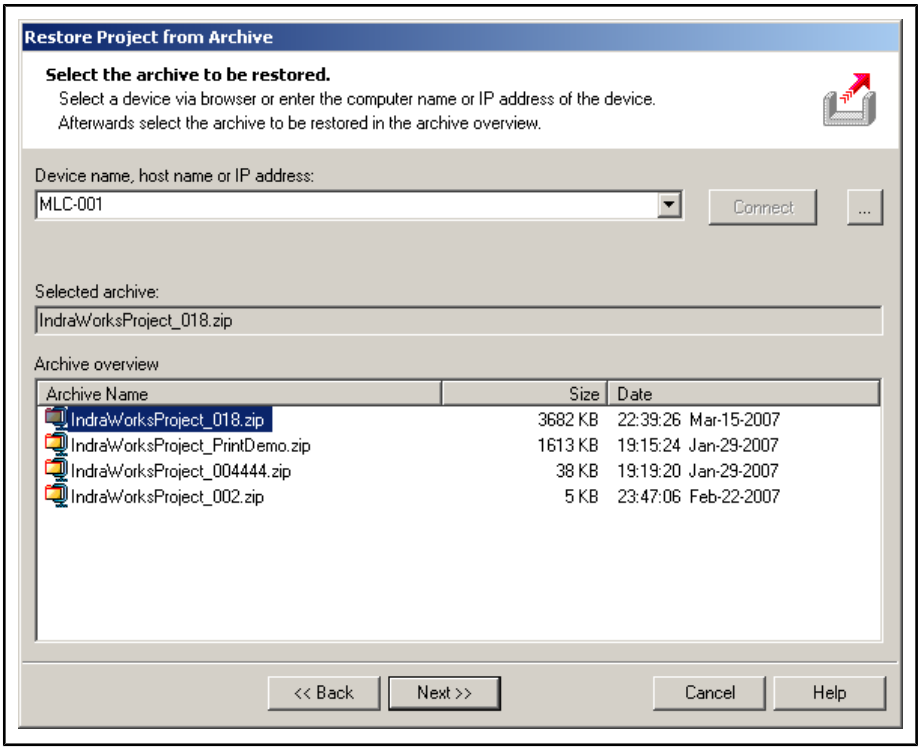


Fig.2-37: Restoring a project from a device; selecting the archive

Device Name, Host Name or IP Address

Enter the device containing the archive to be restored in the “Device name, host name or IP address” input box. You can do this in four ways:

- Enter the IP address (nnn.nnn.nnn.nnn)
- Enter the computer name of the destination device
- Select the destination device via a drop-down list. This list box contains all FTP capable devices of the active project as well as the five destination devices (device name, host name or IP address) last used in restoring.
- Select a device via the “…” browser button.

If you select the device from the drop-down list or the device browser, the connection to the selected device is established automatically.

If you enter the IP address or the computer name, establish the connection to the destination device by “Connect”.

After the connection has been established, all archives available on the device are displayed in the “Archive overview” list. Select the archive to be restored and click “Next>>”.

Selecting the Destination Directory

Select here a directory of the local drive, where you want to restore the project from the archive.

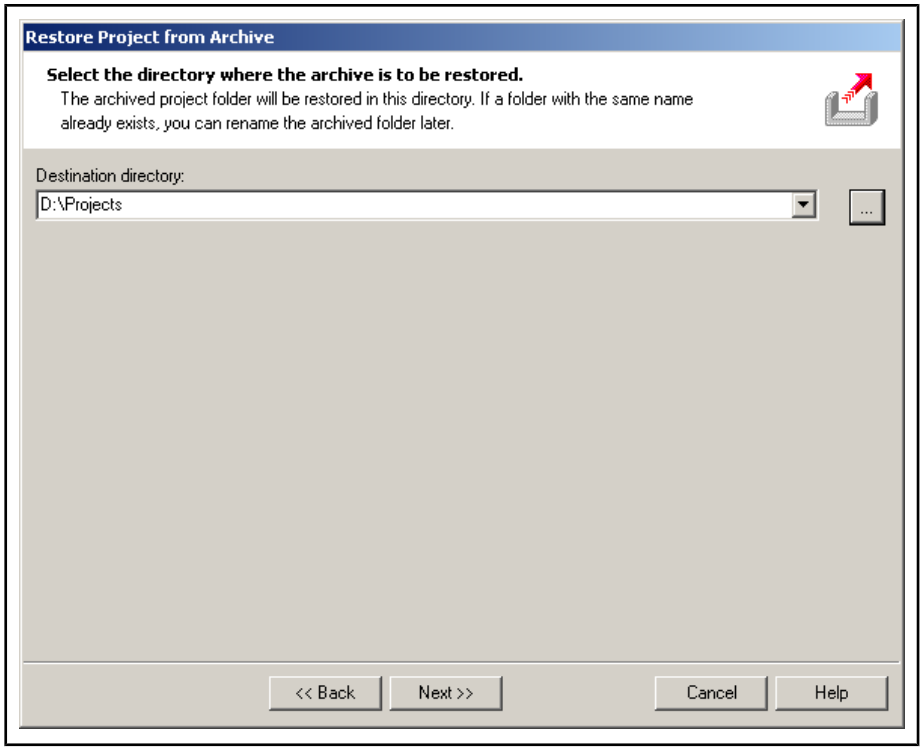


Fig.2-38: Restoring a project from a device; selecting the destination directory

Checking the User Entries

Here you can check your settings. Start restore by the “Finish” button.

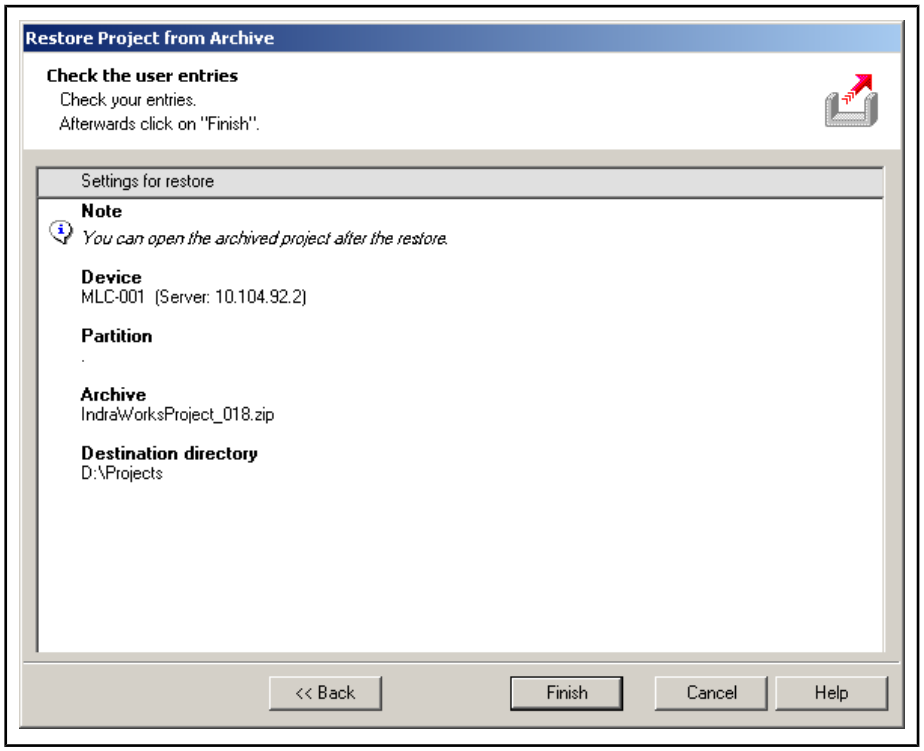


Fig.2-39: Restoring a project from a device; check of entries

1. Progress Bar “Get Archive from Destination Device”

During restore, the archive is copied from the device to the local drive first. This process is shown in a progress bar.

Entering the Password

If you have protected the archive with a password, you will now be prompted to enter that password.

2. Progress Bar “Restore on Temporary Directory”

Then the project is restored from the archive to a temporary directory of the local drive. This process is shown in a progress bar. After restore, the project is copied to the destination directory.

If the selected destination directory already exists, you will be prompted to enter a new one.

Summary

After restoration has been completed, settings and results are displayed.

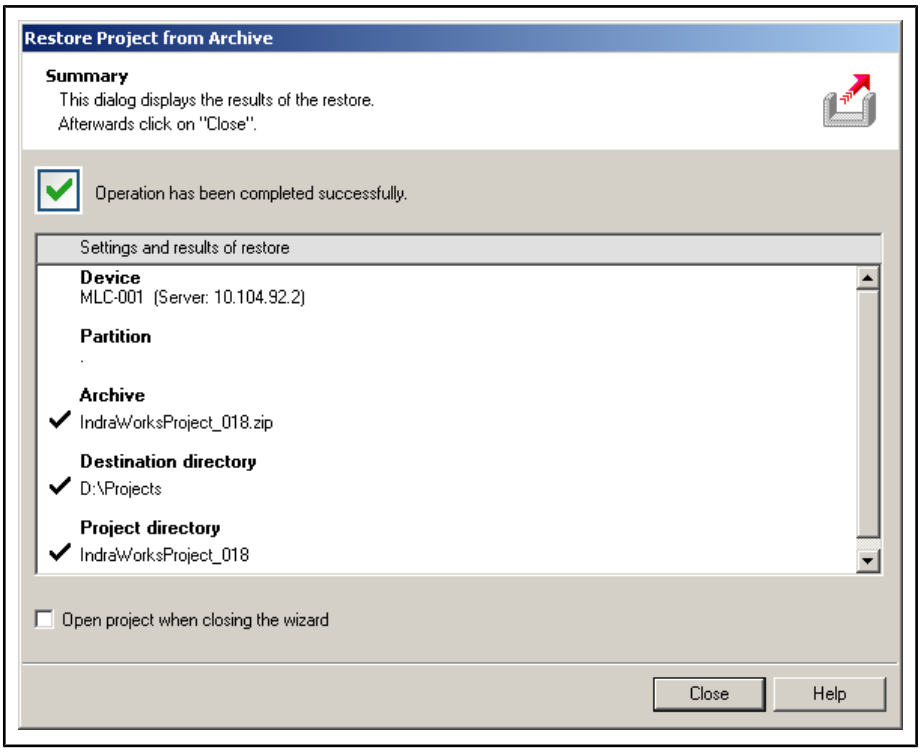


Fig.2-40: Restoring a project from a device; results

2.2.6 Archiving and Restoring Workspaces

General Information

In IndraWorks you can archive the open workspace together with all its projects and restore it later. Just like when archiving projects, you can use archives on the file system or on an FTP server.

Archiving a Workspace

To start the archiving wizard use the **Archive…** entry in the context-sensitive menu of the workspace node.

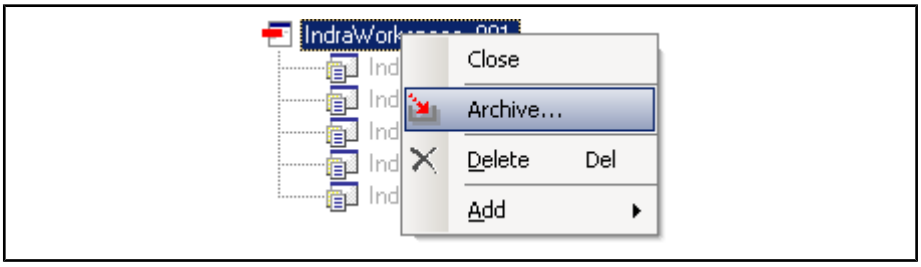


Fig.2-41: Archive a workspace, context-sensitive menu

Select the settings for the archive and click on “Next>>” till the wizard displays the page for checking these settings. The section “Project” contains the workspace file as well as all of the included projects to be archived:

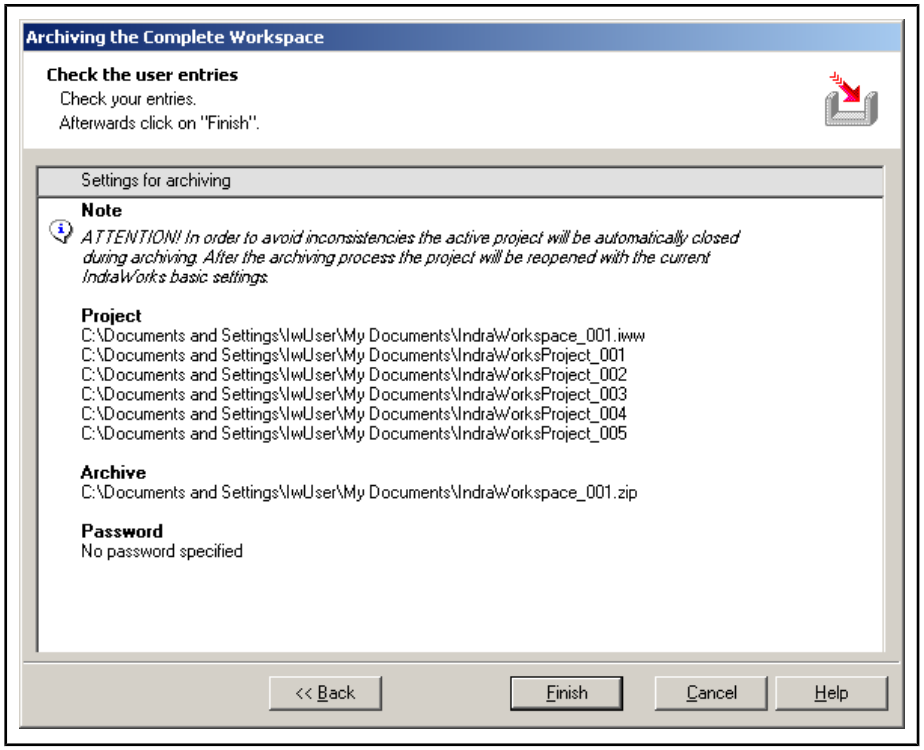


Fig.2-42: Archiving the Complete Workspace dialog box

To create the archive select “Finish”. The summary displays the result of the archiving:

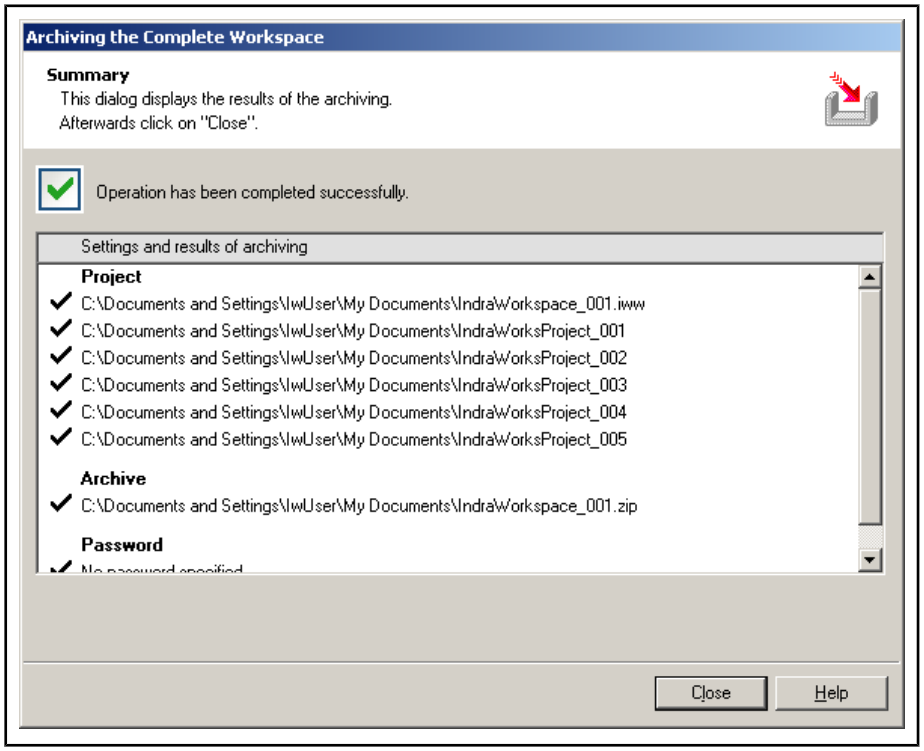


Fig.2-43: Result of the archiving

☞ If the workspace or a project could not be archived, an error symbol will be displayed.

Restoring a Workspace

To restore a workspace and its projects from an archive, use **Project ▸ Restore…** and select the desired archive on the file system or on an FTP server.

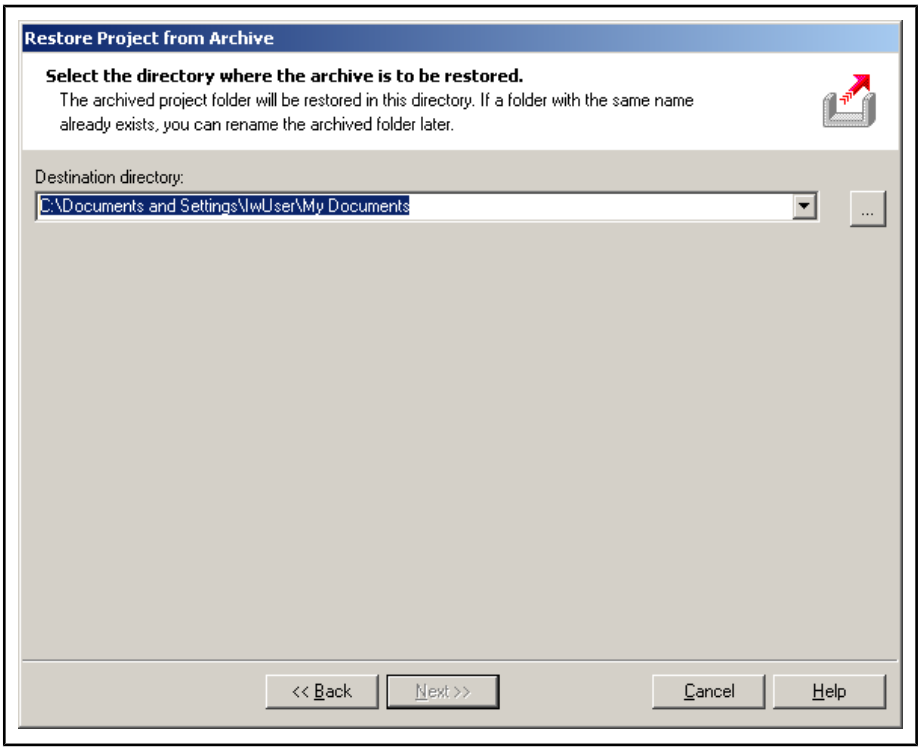


Fig.2-44: Restore Project from Archive dialog box

Select the destination directory and click on “Next>>” and “Finish”. Now the archive will be restored in the destination directory.

If the workspace or projects of the archive already exist in the destination directory, you can define whether they will be overwritten.

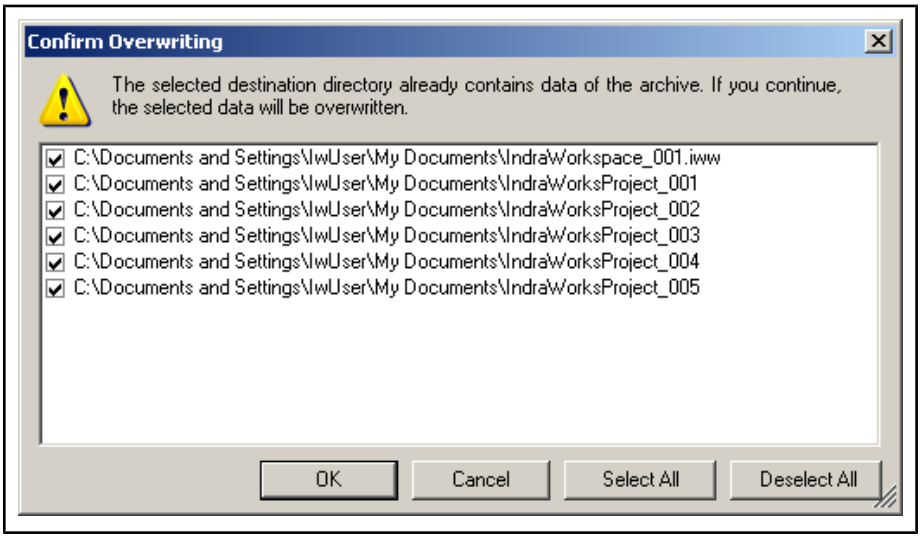


Fig.2-45: Overwrite projects

If a project to be overwritten is open, IndraWorks displays an error message when “OK” is pressed:

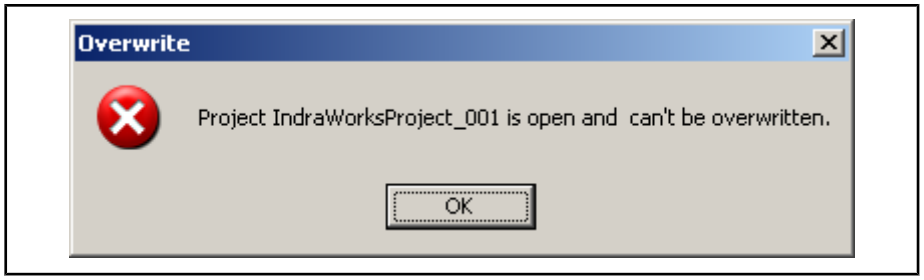


Fig.2-46: Overwirte error message

In this case deselect the corresponding project and press “OK” again, or cancel the operation and close the open project.

Select “OK” to display the result of the restore.

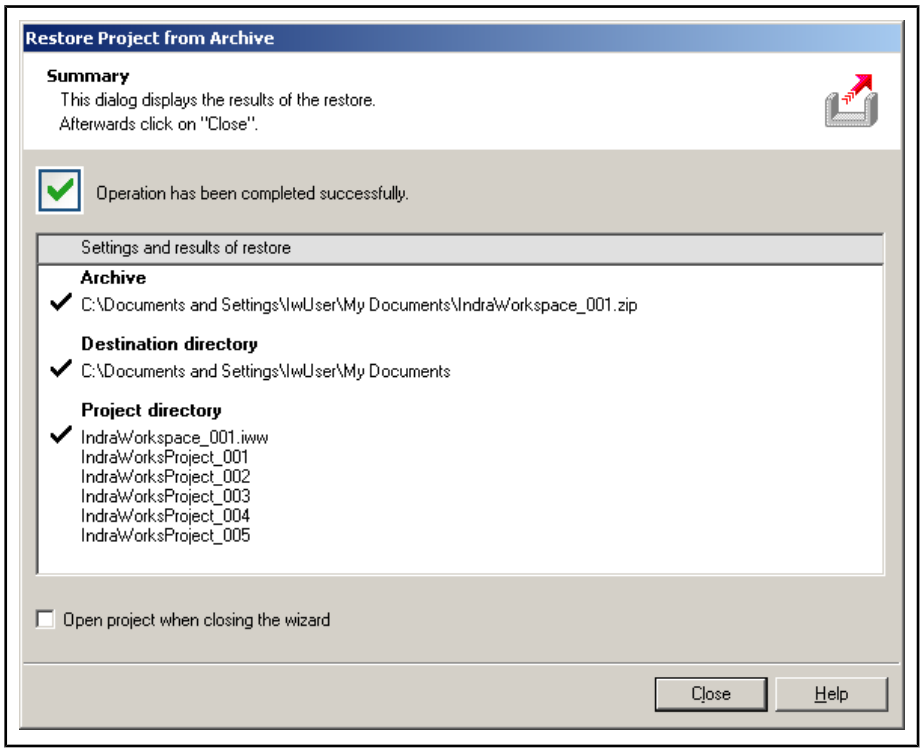


Fig.2-47: Result of the restore

The section “Project directory” displays the restored data. If you select the option “Open project when closing the wizard”, the workspace will be opened and displayed in the project explorer.

2.2.7 Activating the Project for the Operation Desktop

IndraWorks Operation can only be started with an activated project.

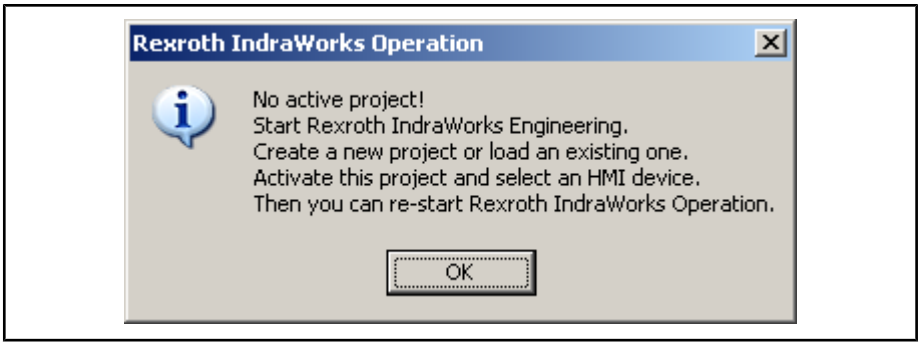


Fig.2-48: Rexroth IndraWorks Operation message window

To activate the project, proceed as follows:

1. Start the Engineering Desktop and open the project you want to activate for the Operation Desktop.
2. Execute an HMI download for the HMI device (e. g. BTV 40) available in the project.
3. Activate the project via **Project ▸ Activate for IndraWorks Operation**.
4. Exit the Engineering Desktop.
5. Start the Operation Desktop.

The Operation Desktop will now visualize the activated project.

☞ This function is available, only if you have installed the IndraWorks LOGIC system or the IndraMotion MTX system.

2.2.8 Scan for Devices

General Information

In IndraWorks you can scan for available devices of the installed libraries. This option is supported by a wizard.

Depending on the device type selected, you can scan the devices via the serial interface or the Ethernet, PCI or Profibus interface.

To call the scan wizard, select an empty project, switch to the online mode and select **Scan for devices…** from the context-sensitive menu.

The device is switched to the online mode automatically, if you select **Project ▸ Scan for Devices...** or the following button from the toolbar:

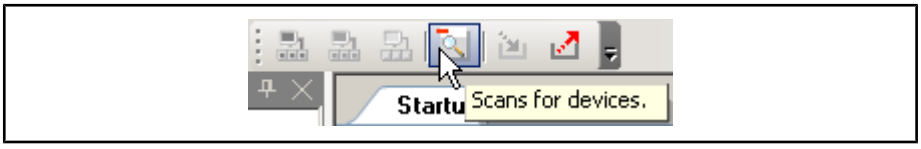


Fig.2-49: Project toolbar, scan for devices

If you open a page of the wizard for the first time, the input boxes will contain default values. Otherwise, those values will be displayed that you have entered last for a successful scan for a device and for an application to a project.

If the entered values are correct, you can move between the pages of the wizard by “<<Back” and “Next>>”.

The last page, the scan dialog, provides the “Finish” button which is active once a device has been found. This button stops the scan process; the found devices are applied to the project. Any values you have entered on the wizard pages will be saved for future scanning.

You can exit the wizard at any time with “Cancel”. Changed values will not be saved, and the scan process is canceled.

Operation

Select Devices

First select the device types you wish to scan for. Select the appropriate items in the “Installed” list and apply them by “right arrow”, by double-click or drag-and-drop. Selected devices can be deselected by “left arrow”, by double-click or drag-and-drop. The scan order can be changed by “arrow up” and “arrow down”.

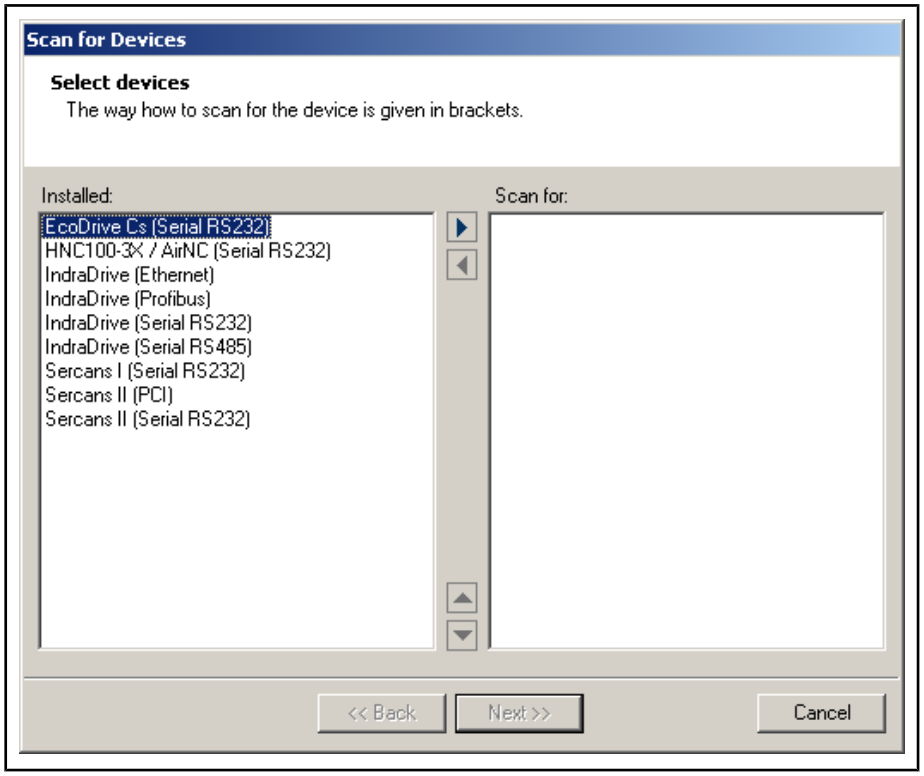


Fig.2-50: Scan for Devices dialog box; selecting the devices

Click on “Next>>” to move to the next dialog. This dialog depends on the interface used to scan for the device. The interface (serial RS232, serial RS485, Ethernet, PCI, Profibus) is specified with the device type.

Settings for the Serial RS232 Interface

If you have selected a device type allowing the scanning via a serial RS232 interface, the following dialog will be displayed.

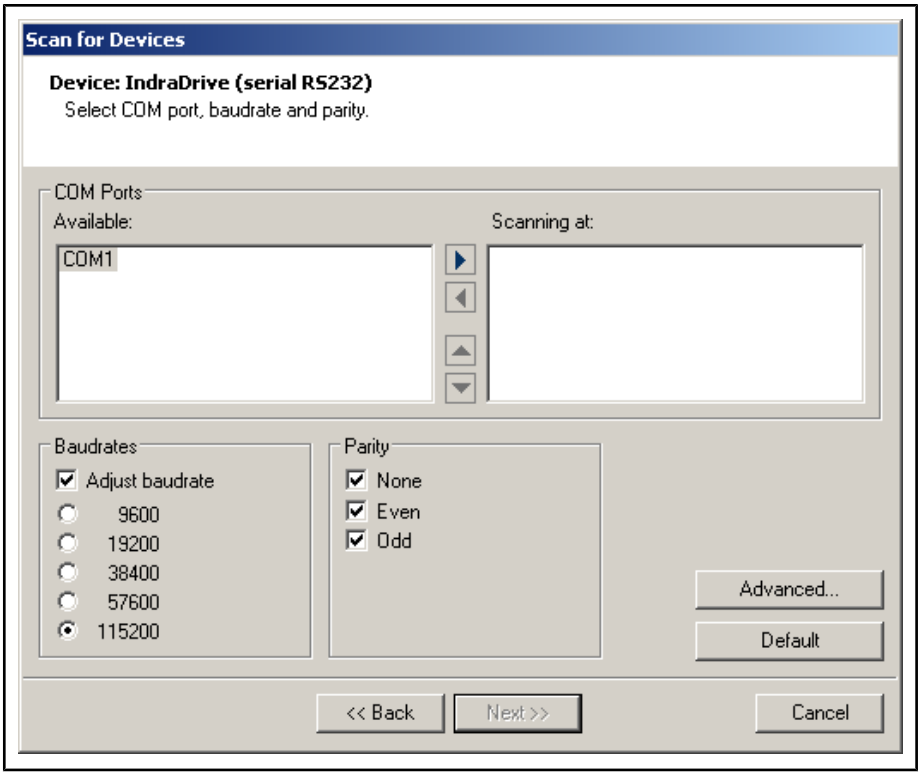


Fig.2-51: Scan for Devices dialog box, serial RS232 device data

To define the COM ports to be used for scanning for a device, apply the appropriate items of the “Available” list by “right arrow”, double-click or drag-and-drop. Selected COM ports can also be deselected. The order can be changed by “arrow up” and “arrow down”.

In addition, the parity and the baudrate must be selected for the scanning. If the check button “Adjust Baudrate” is activated, the scanner will try and adjust devices connected at this interface to the selected baudrate.

☞ Adjusting the baudrate can change the baudrate of devices already configured. Thus the communication with those devices may be disturbed. Please adjust the configuration of those devices to enable a communication!

Click “Advanced...” to open a dialog box where you can enter the timeout for the device type used by the scanner.

☞ The scanner attempts to establish a communication connection to the devices to be scanned. The timeout is the time interval available to the connected devices for responding to a request of the scanner, thus identifying themselves. The scan process is decelerated by a high timeout and accelerated by a low timeout. If the timeout is too low, the scanner might fail to detect all devices.

☞ We recommend to use the default timeout value.

To reset the changed values to the default values of the library, click on “Default”.

If you have selected COM port, baudrate and parity, the “Next>>” button is activated. Use this button to move to the next setting dialog or to the scan dialog.

Settings for the Serial R485 Interface

In order to address several devices via a serial interface, it is possible to construct a RS485 bus by RS232/RS485 converters.

Basically, the scanner settings for the serial RS485 interface correspond to the settings for the serial RS232 interface. Only the differences between RS485 interface and RS232 interface will be explained here.

If you have selected a device type allowing the scanning via a serial RS485 interface, the following dialog will be displayed.

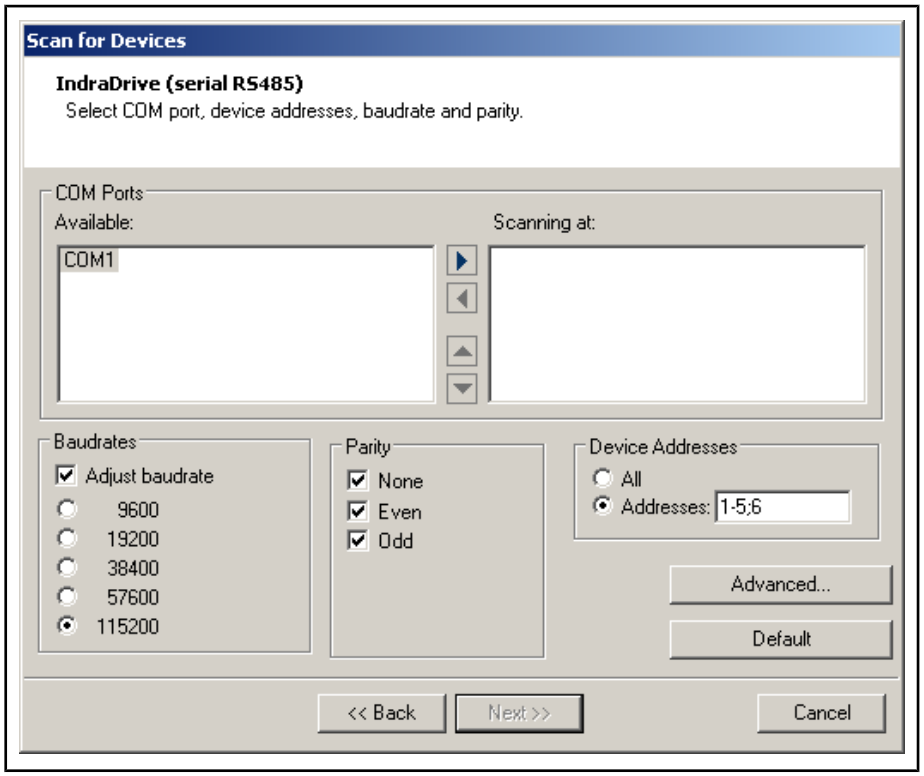


Fig.2-52: Scan for devices, serial RS485 device data

In contrast to the RS232 interface you have to enter here a scan area for the device addresses. The allowed area ranges from 1 to 127. If you enter single addresses, separate them by semicolon. To specify address areas enter a start address, a hyphen and an end address. Use “Advanced...” to open a dialog for further settings.

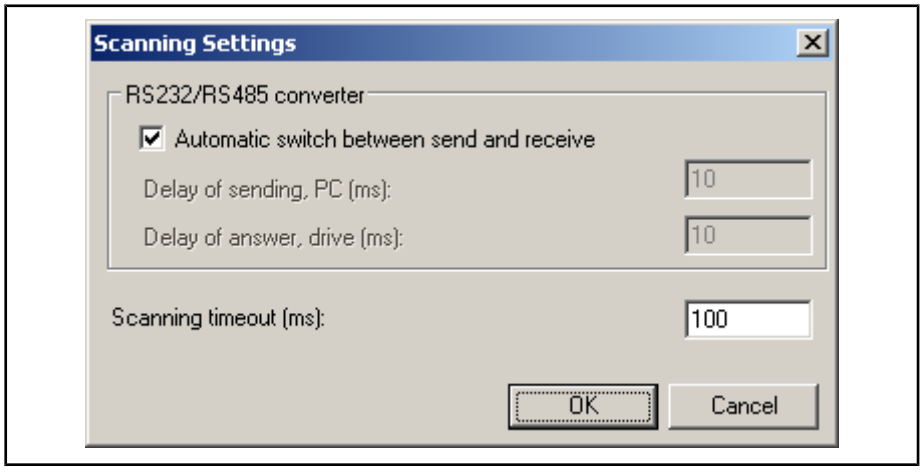


Fig.2-53: Scan for devices, serial RS485 device data

“Automatic switch between send and receive”

If RS232/RS485 converters with automatic switch between send and receive (recommended) are used, the settings for “Delay of sending, PC” and “Delay of answer, drive” are irrelevant and thus disabled.

If a converter is used, that needs signals to switch between send and receive, you have to enter a delay time for sending in the PC and a delay time for answer in the drive, which ensure a reliable communication connection.

☞ If converters without automatic switch between send and receive are used, the FIFO buffer of the according serial interface must be disabled. For information on this, please refer to the documentation of the operating system.

☞ You can get further information on RS485 in the documentation of the drive setup.

Settings for the Ethernet Interface

If you have selected a device type allowing the scanning via an Ethernet interface, the following dialog box will be displayed.

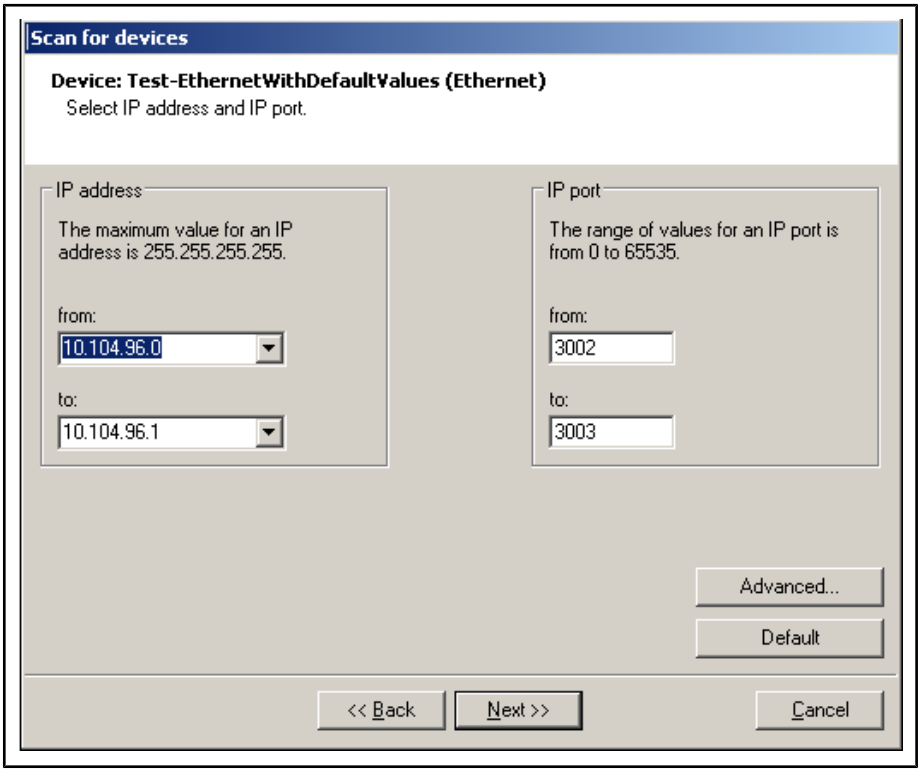


Fig.2-54: Scan for Devices dialog box, Ethernet device data

You must enter at least one IP address and one IP port. But you can also enter ranges. In this case, the IP addresses may only differ from each other in the final numerical area. The input boxes for the IP addresses contain a drop-down list with the recent ten valid IP addresses. Open this list by <F4> or click on “down arrow” and select an IP address.

If you enter invalid characters, a red circle with a white exclamation mark will be displayed when you exit the input box. The tool tip for this icon shows the cause of the message.

Click “Advanced…” to open a dialog where you can enter the timeout for the device type.

☞ The scanner attempts to establish a communication connection to the devices to be scanned. The timeout is the time interval available to the connected devices for responding to a request of the scanner, thus identifying themselves. The scan process is decelerated by a high timeout and accelerated by a low timeout. If the timeout is too low, the scanner might fail to detect all devices.

☞ We recommend to use the default timeout value.

To reset the changed values to the default values of the library, click on “Default”.

When you have selected an IP address and an IP port, click on “Next>>” to move to the next setting dialog or to the scan dialog.

Settings for PCI

If you have selected a device type allowing the scanning via a PCI interface, you do not have to enter any further data. The next setting dialog or the scan dialog will be opened directly.

Settings for Profibus

If you have selected a device type allowing the scanning via a Profibus, the following dialog will be displayed.

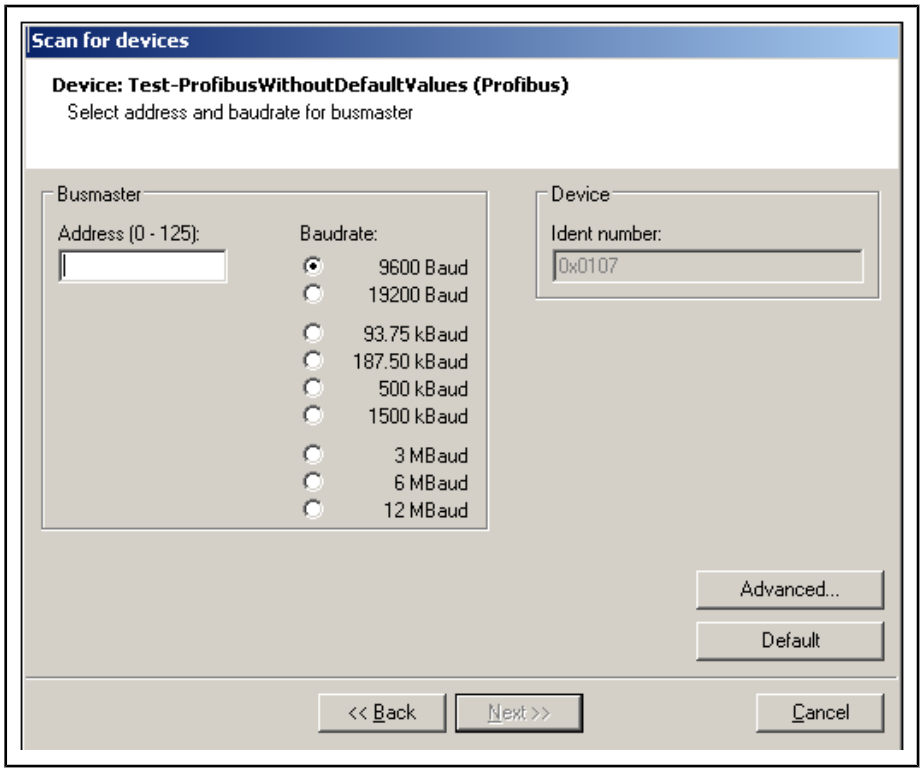


Fig.2-55: Scan for Devices dialog box, Profibus device data

Enter a baudrate and an address for the Profibus master.

The ident number of the device type is displayed, but cannot be edited.

If you enter invalid characters or a number outside the range of value, a red circle with a white exclamation mark will be displayed when you exit the input box. The tool tip for this icon shows the cause of the message.

Click “Advanced...” to open a dialog box where you can enter the timeout for the device type used by the scanner.

☞ The scanner attempts to establish a communication connection to the devices to be scanned. The timeout is the time interval available to the connected devices for responding to a request of the scanner, thus identifying themselves. The scan process is decelerated by a high timeout and accelerated by a low timeout. If the timeout is too low, the scanner might fail to detect all devices.

☞ We recommend to use the default timeout value.

To reset the changed values to the default values of the library, click on “Default”.

If you have selected an address and a baudrate for the bus master, click on “Next>>” to move to the next setting dialog or to the scan dialog.

Scan Dialog

When you have edited the setting dialog for each selected interface, the scan dialog will open. This also starts the scan process automatically. You can stop this process at any time by “Stop scanning for devices” and restart it by “Start scanning for devices”.

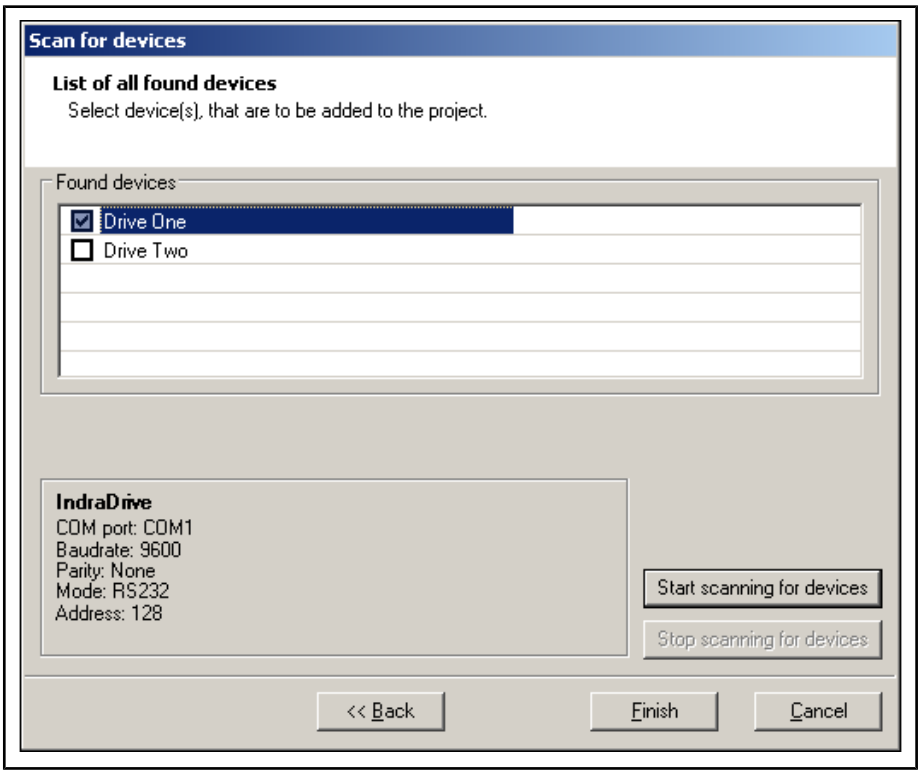


Fig.2-56: Scan for Devices dialog box, Found devices

Editing and Applying Found Devices

Once a device has been found, it will be displayed in the list and the “Finish” button will be activated.

To display additional information, just click on a device in the list.

Activate the check box to define that a device is to be added to the project.

Click on “Finish” to exit the scan process and to apply the found devices selected to the project. Depending on the device type, another wizard allowing the entry of further parameters may be started when the devices are applied to the project.

To rename a found device, press <F2> or use the mouse.

2.2.9 Offline and Online Mode

Offline Mode

When a new project is created or when an existing project is opened, the connected devices are in the offline mode. Any changes of project data will be saved to the project, but they will not have any effect on the data in the destination devices.

Offline Mode Display

The offline mode of a device can be recognized in the display of its node in the project tree:

- no mode symbol on the icon

Fig.2-57: Device node, offline mode

Online Mode

In the online mode, your changes will be saved to the project and simultaneously be transferred to the data memory of the destination device.

CAUTION

Caution

Material damage may be caused by failures in the activation of motors and moving elements!

⇒ In the online mode, considerable material damage or personal injury may be caused by an inadvertent transfer of data to the destination device. Before you change any project data, please make absolutely sure, that this change will not cause any damage or injury.

Online Mode Display

The online mode of a device can be recognized in the display of its node in the project tree:

- blue mode symbol on the icon
- blue font

Fig.2-58: Device node, online mode

2.2.10 Switching Between Offline and Online

What Happens During the Switch

Four Switching Steps

Once you switch a device to the online mode, IndraWorks performs the following steps:

- First, IndraWorks attempts to establish a communication connection to all devices in the project (communication adjustment).
- Then, IndraWorks checks whether the projected device structure is equal to the existing device structure (structure adjustment).
- In the third step, IndraWorks checks whether the projected device configuration is equal to the existing configuration (configuration adjustment).
- In the final step, IndraWorks compares the data of each device in the project with the appropriate data in the destination device (data adjustment).

A device will not be switched to the online mode before all the above steps have been completed successfully.

The sections below will provide a detailed description of these steps.

How to Proceed

Switch a project to the online mode by **Project ▸ Switch Devices Online...**. You can also use **Switch Online** in the context-sensitive menu of the device node.

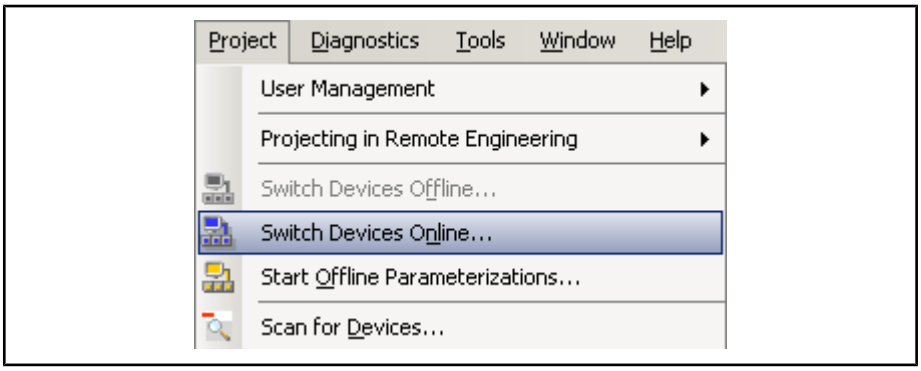


Fig.2-59: Switching to the online mode

If several devices are connected, you will be prompted to select the devices for the online edit:

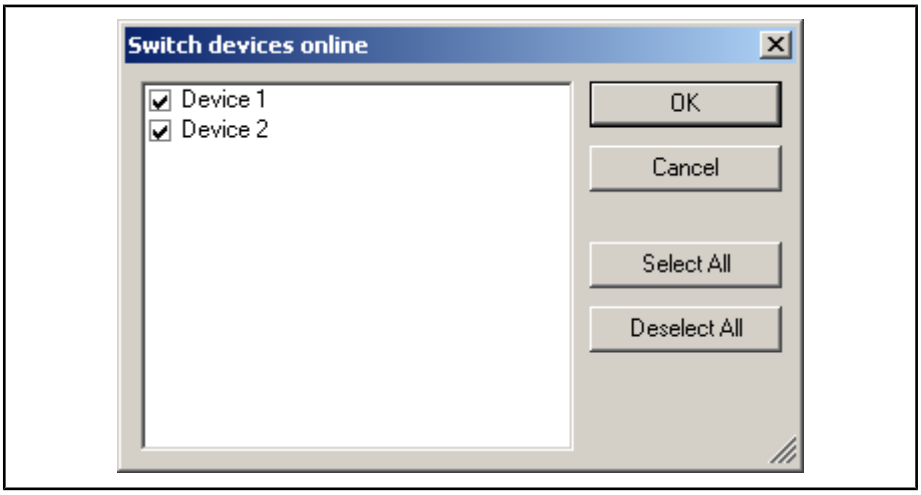


Fig.2-60: Remote Engineering Select devices for online switch

Communication Adjustment

Checking the Communication Connection

First, the communication connection to the devices of the project is verified. If IndraWorks can communicate with all devices, the structure adjustment is automatically carried out in the next step. Otherwise, you can

- scan for a device,
- or retry switching this device to the online mode,
- or stop switching this device to the online mode.

Fig.2-61: Switching to the online mode, communication connection message

Scanning for a Device

The scanner is used for searching a device (see chapter 2.2.8 "Scan for Devices" on page 40)

If the scanning is successful, the communication settings of the device are applied. If the scanning is not successful, switching the device to the online mode is stopped.

Structure Adjustment

In the structure adjustment step, IndraWorks checks whether the projected device structure is equal to the existing structure. E. g., the device peripherals for a control are checked as follows:

- number and type of the drives connected,
- number and type of the connected I/O bus devices.

The structure adjustment depends on the device type. You can find details on switching a device to the online mode in its documentation.

If the projected structure is equal to the existing structure, the configuration adjustment is carried out in the next step.

Eliminating Structure Inconsistencies

If the data in the project differ from the data in the destination device, they will be shown in a tool window. The tool window provides the following information:

- overview of structure inconsistencies,
- data set in the project,
- data existing in the destination device,
- reason for the problem,
- possible solution to the problem.

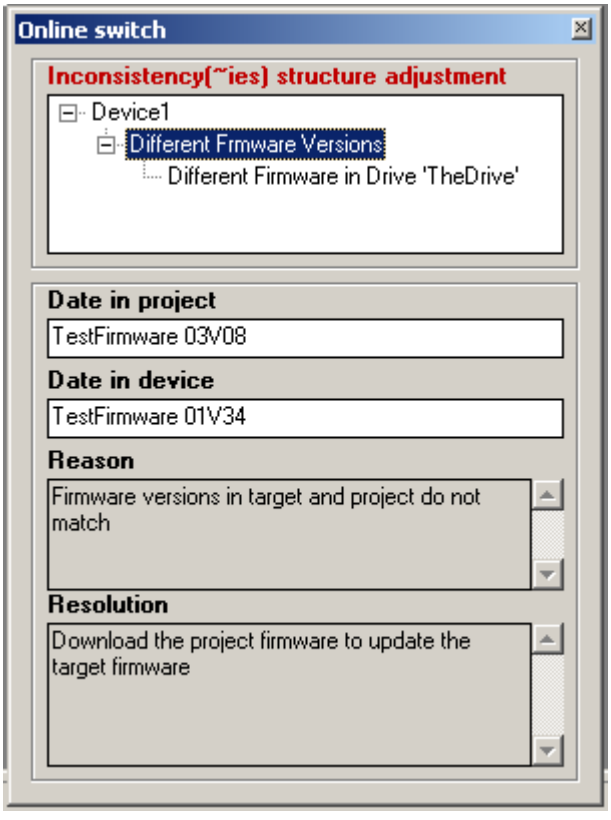


Fig.2-62: Online switch, information on structure inconsistencies

Once you have eliminated the inconsistencies, you can restart the switch. If IndraWorks does not detect any inconsistencies in the structures any longer, the configuration adjustment is carried out as the next step.

Configuration Adjustment

The configuration of the various devices must be adjusted (e. g. the projected drive number may differ from the actually set drive number in a Sercos ring).

☞ The configuration adjustment depends on the device type. You can find details on switching a device to the online mode in its documentation.

If the projected configuration is equal to the existing configuration, the data adjustment is carried out in the next step.

Eliminating Configuration Inconsistencies

If the data in the project differ from the data in the destination device, they will be shown in a tool window. The tool window provides the following information:

- overview of configuration inconsistencies,
- data set in the project,
- data existing in the destination device,
- reason for the problem,
- possible solution to the problem.

Once you have eliminated the inconsistencies, you can restart the switch. If IndraWorks does not detect any configurations inconsistencies any longer, the data adjustment is carried out in the next step.

Data Adjustment

The data of the various devices must be adjusted, e. g.:

- motion program of a control,
- velocity limitation of a drive.

☞ The data adjustment depends on the device type. You can find details on switching a device to the online mode in its documentation.

Eliminating Data Inconsistencies

If the data in the project differ from the data in the destination device, they will be shown in a tool window. The tool window provides the following information:

- overview of data inconsistencies,
- data set in the project,
- data existing in the destination device,
- reason for the problem,
- possible solution to the problem.

Once you have eliminated the inconsistencies, you can restart the switch. If IndraWorks does not detect any data inconsistencies any longer, switching to the online mode is completed.

2.3 Adding Files to an IndraWorks Project

2.3.1 General Information

Existing files can be added to an IndraWorks project, so that documentations, data sheets or additional information will be transferred with the project.

The embedded file is displayed in the project. To edit the file, call the currently registered application in IndraWorks.

☞ If **Automation Interface** is used, you can add or run HTML sites with embedded script code, providing the access functionality to the Automation Interface, in an IndraWorks project.

2.3.2 Adding a File

If a project node allows the insertion of files, its context-sensitive menu provides **Add ▸ File**.

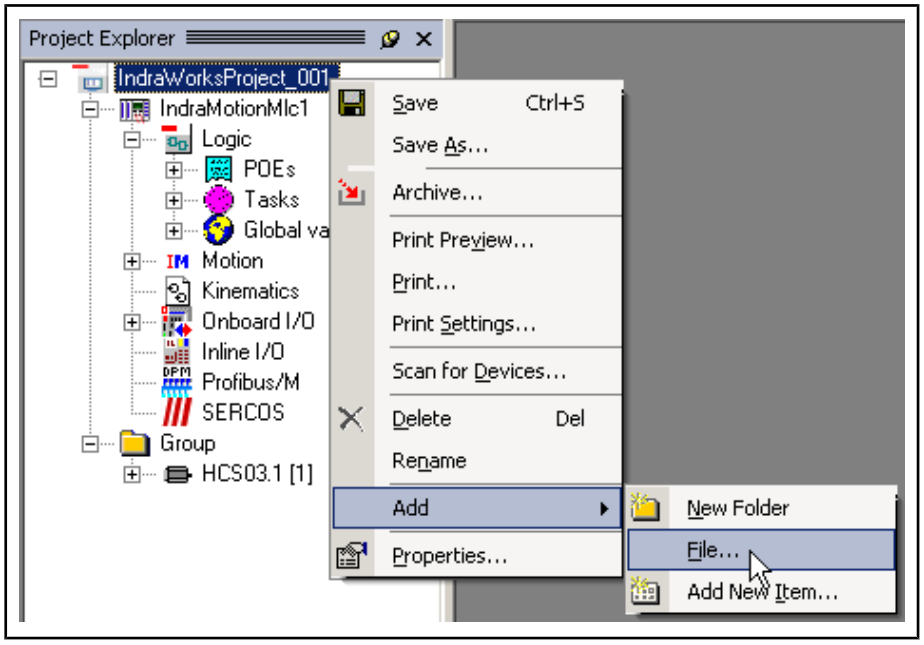


Fig.2-63: Adding files to a project

Selecting this menu item opens the “Open” dialog box.

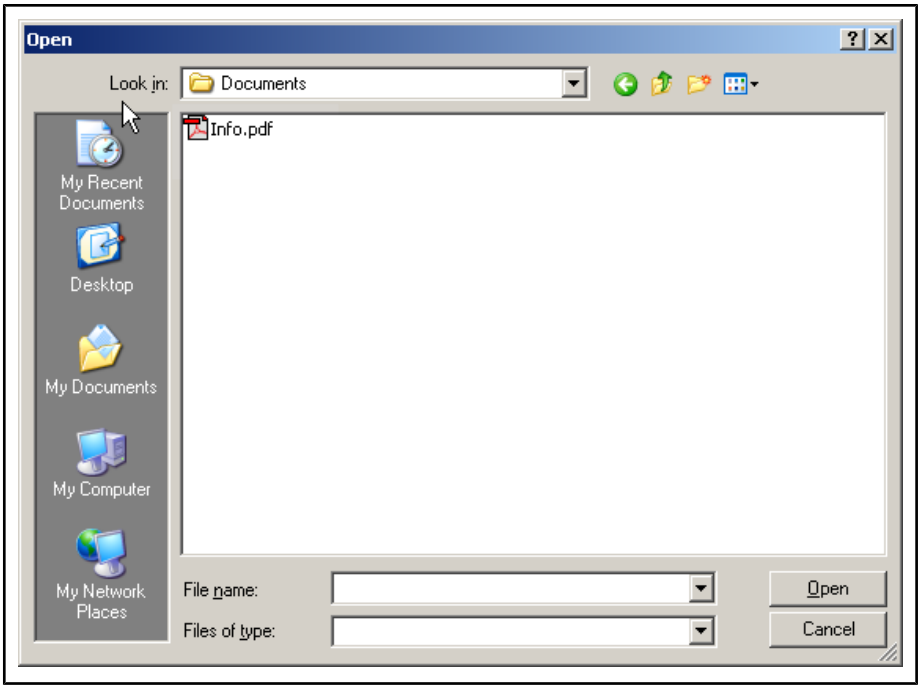


Fig.2-64: Open dialog box

Select a file and insert a copy of this file in your project by clicking on the “Open” button. The file can then be edited in your project.

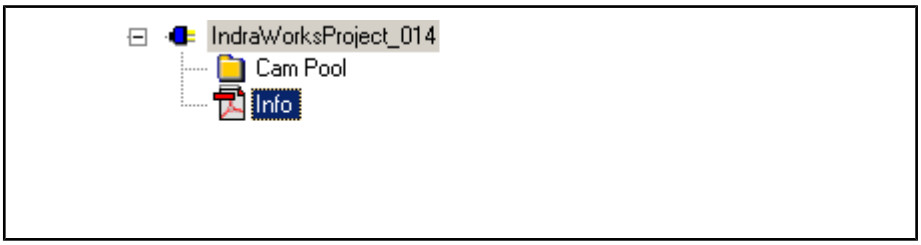


Fig.2-65: File added to the project

The icon used is the default icon of this file type. The name is equal to the file name without filename extension.

2.3.3 Editing a File

Open

To open the embedded file, double-click on it, press <Enter> or use the context-sensitive menu. IndraWorks will start the application registered for editing in the Windows operating system.

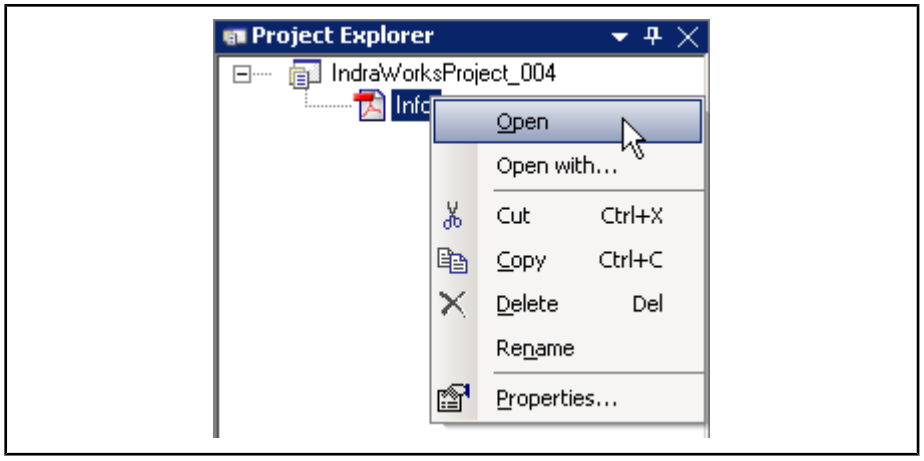


Fig.2-66: Editing an embedded file

In the case shown above, Acrobat Reader (PDF) will be started.

IndraWorks can start the editing application when the file is opened and exit it when the project is closed or the user interface is shut down.

All other actions, such as printing or saving a file, are executed by the editing application.

Open with… If there is no registered application or if you wish to register a different application, select **Open with…** from the context-sensitive menu.

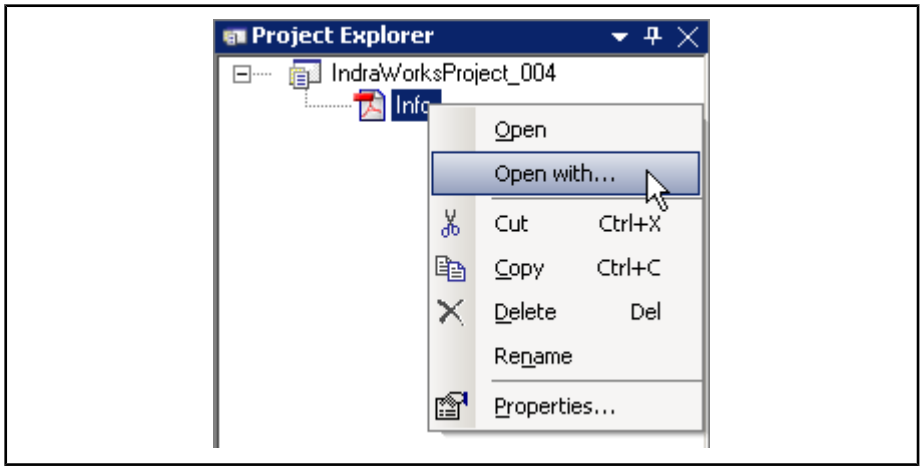


Fig.2-67: Embedded file, Open with…

Select an application for editing your file in the “Open with” dialog box. If you activate the check box “Always use the selected program to open this kind of file”, this application will be used for all further editing actions.

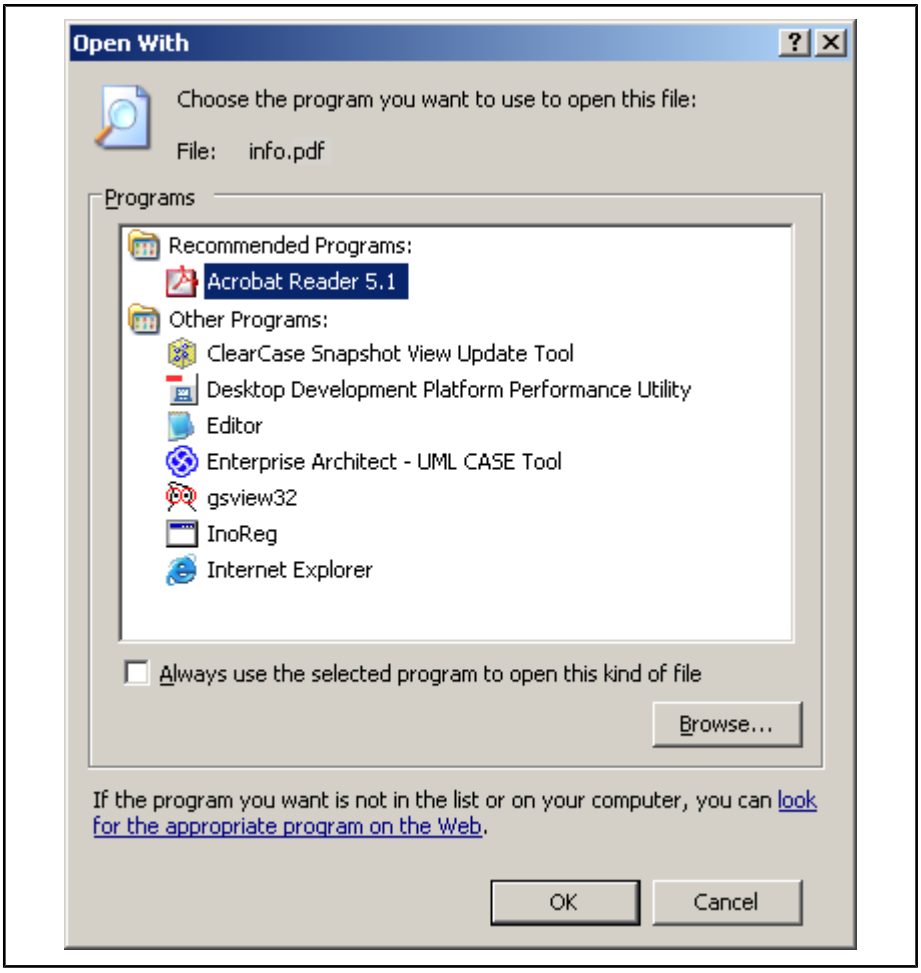


Fig.2-68: Open with dialog box

This dialog does not start modally, i. e. if you click on the IndraWorks desktop while this dialog box is open, it will disappear to the background. To put it to the foreground again, press <Alt>-<Tab>.

Automation Script... Having added an HTML page with embedded Automation Script to the project, the context-sensitive menu will provide **Automation Script...** Use this menu entry to obtain access to the properties of the Automation Script.

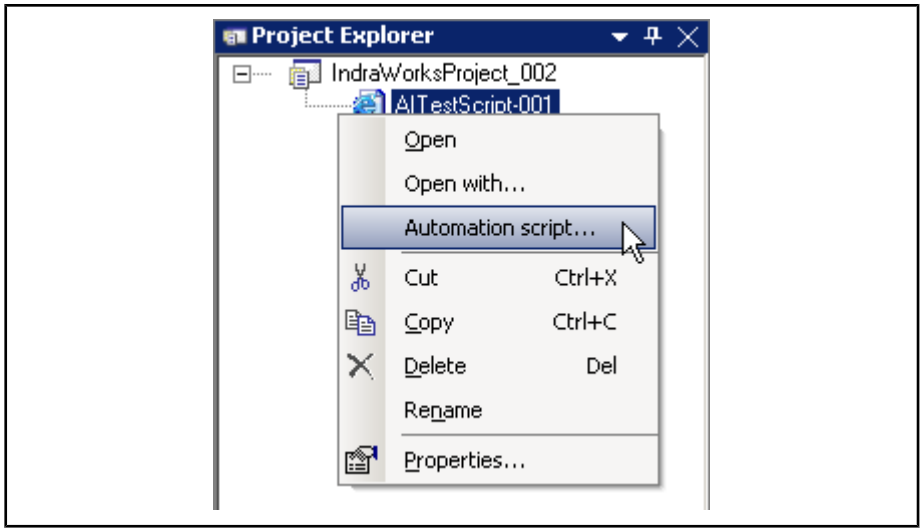


Fig.2-69: Embedded file, Automation Script...

Selecting this menu item opens the “Automation Script Configuration” dialog box.

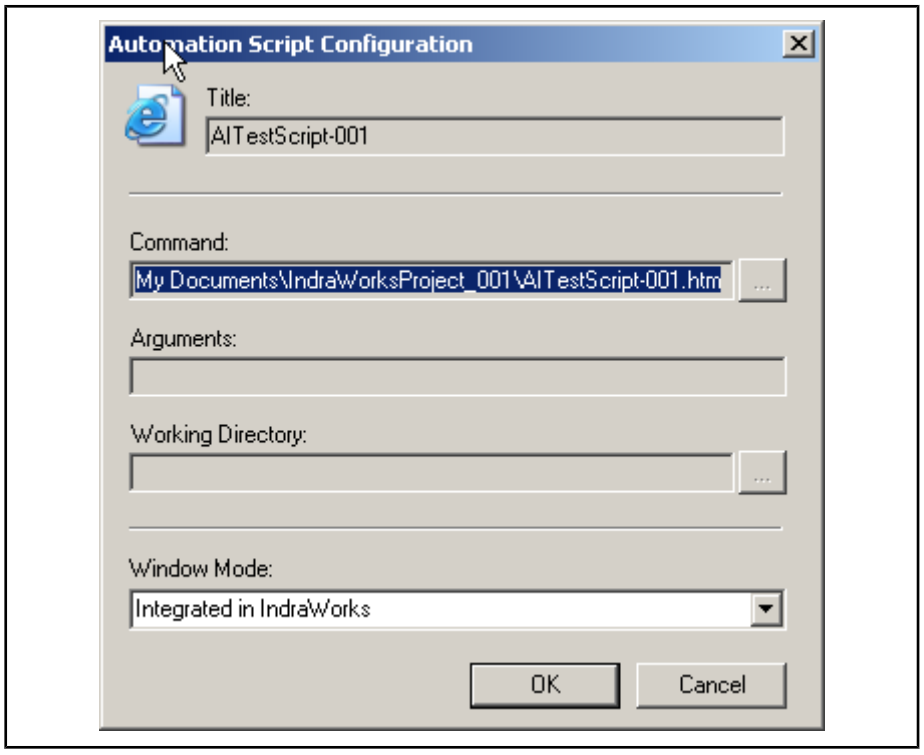


Fig.2-70: Automation Script Configuration dialog box

The “Automation Script Configuration” dialog box displays the file name and the filing location of the HTML file. In the Window mode list, you can define whether the HTML page is to be started integrated in IndraWorks or externally.

The current version starts HTML pages only integrated in IndraWorks.

Cutting, Copying, Pasting

To move the file into the project, select **Cut** and **Paste** . You can also use drag-and-drop.

Use **Copy** and **Paste** to create a copy of the file at a different position in the project.

Delete To remove the file from the project, select **Delete**. If started, the editing program will be exited.

Rename To rename the file, press <F2> or use the context-sensitive menu.

Properties Open the “Properties” dialog box via the context-sensitive menu. The dialog box depends on the type of the embedded file.

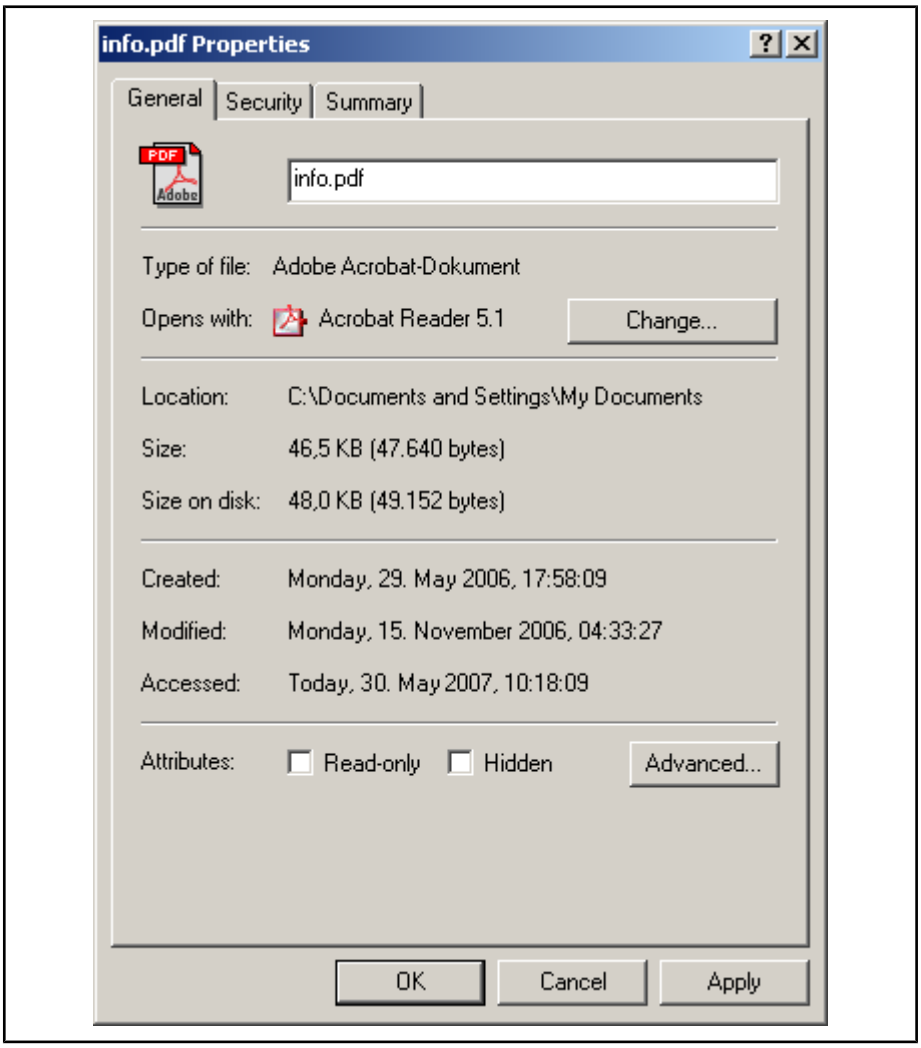


Fig.2-71: Properties dialog box

2.3.4 Adding a New Element from the Templates to the Project

To add a new element to a project select **Add ▸ New Element** in the context-sensitive menu. In the following dialog, first select the category, that contains the desired template. The right part of the dialog box displays the templates available in that category. If you select a template it will be opened by its default editor.

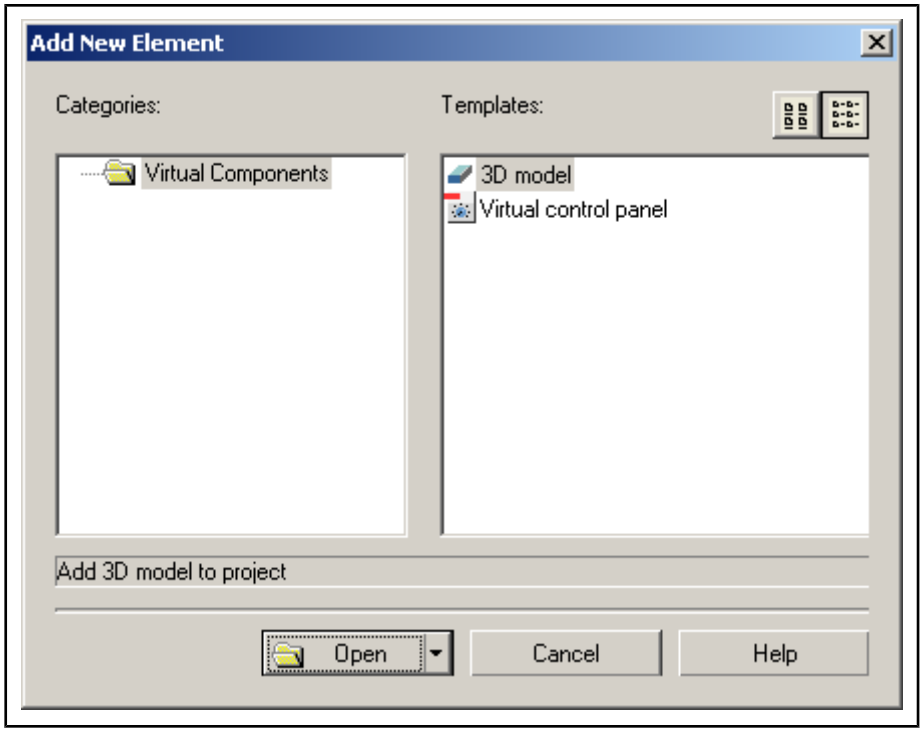


Fig.2-72: Add New Element dialog box

Use the command **Open with** to select another editor for opening this program. You can define this editor as new default editor.

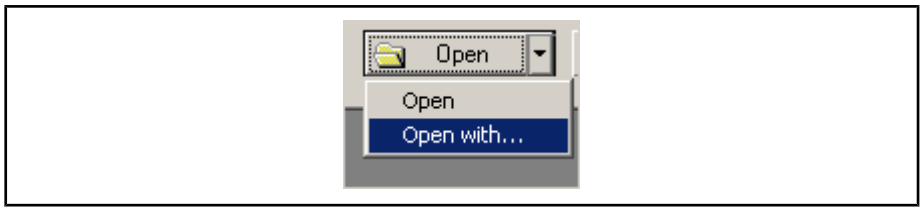


Fig.2-73: Open with another editor

2.3.5 Removing a File from the Project

To remove embedded files from the project, select **Delete**. The subdirectory containing the file will be moved to the Windows recycle bin, together with the file.

2.4 Working with the Workspace

2.4.1 General Information

You can combine and manage several projects in a workspace. It will be displayed in the project explorer and shows the tree structure of the projects and devices.

2.4.2 Workspace Structure

The workspace will be displayed as the root of the tree structure in the project explorer.

The projects desired can be inserted, created and deleted at the levels below the workspace.

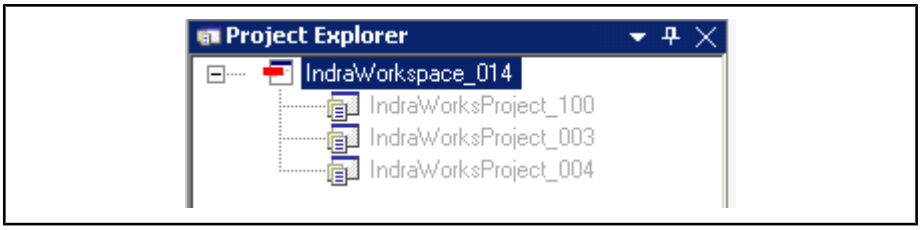


Fig.2-74: Workspace in IndraWorks

2.4.3 Workspace

Creating a Workspace

Projects can only be inserted if a workspace has been created or loaded.

To create a new workspace, select **File ▸ New ▸ Workspace**.

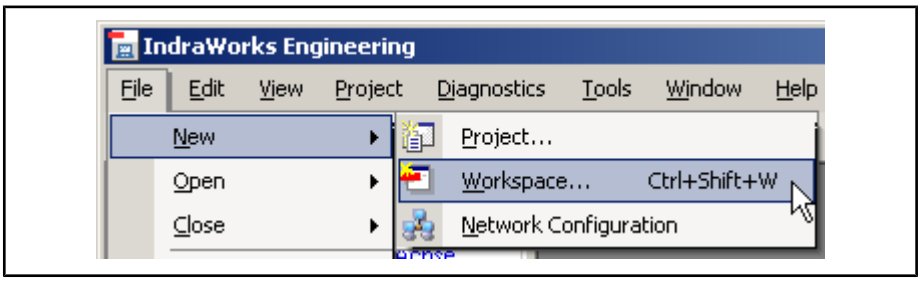


Fig.2-75: Workspace, creating a new workspace

The following dialog box appears:

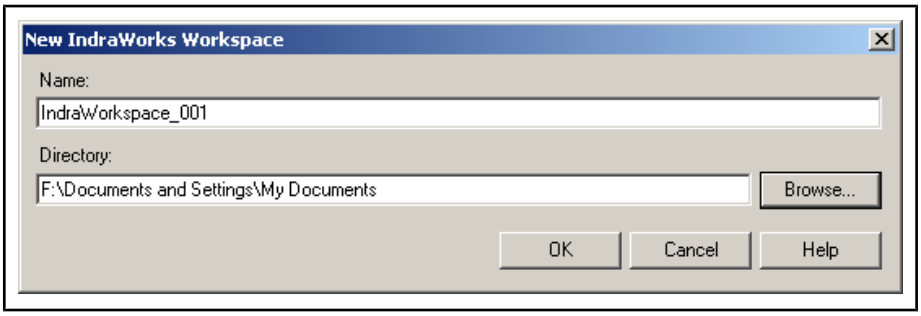


Fig.2-76: New IndraWorks Workspace dialog box

Enter any name and directory for the workspace you wish.

Opening a Workspace

To load existing workspaces, select **File ▸ Open ▸ Workspace**.

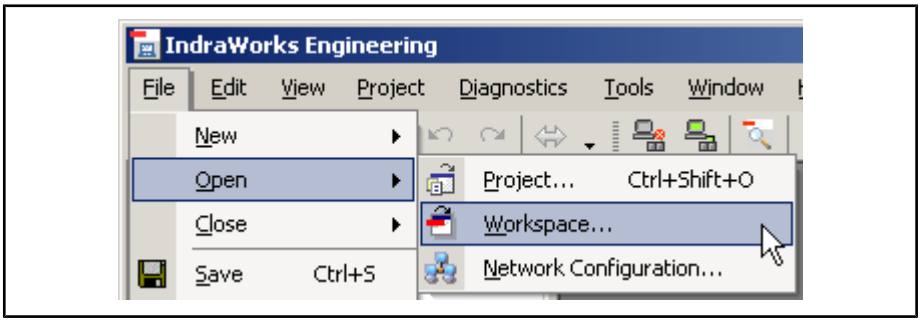


Fig.2-77: Workspace, open a workspace

You can then select a workspace from the following dialog box. The filename extension of the corresponding files is “.iww”.

Deleting a Workspace

The workspace can be deleted via the context-sensitive menu of its node.

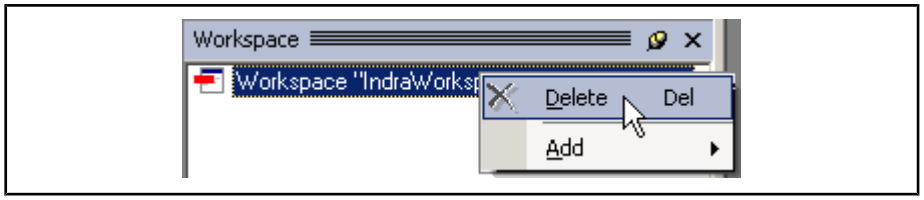


Fig.2-78: Workspace, delete a workspace

This command only deletes the workspace from the data medium, not the projects it contains.

2.4.4 Projects in the Workspace

Adding Projects

The workspace is intended to comprise and manage several projects. There are various ways of adding projects to the workspace:

Integrate an existing project via the context-sensitive menu of the workspace. Select **Add ▸ Existing Project...** The options in the following dialog box are the same as those when projects are opened.

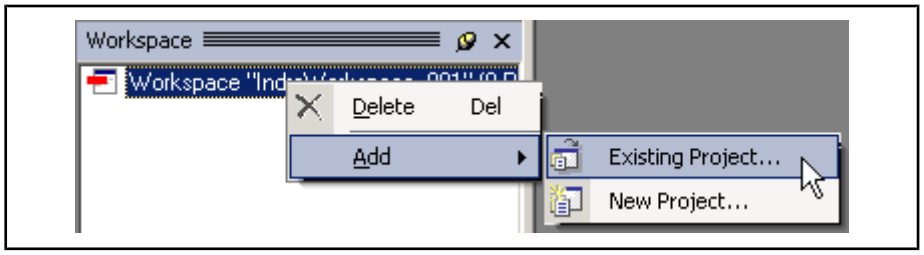


Fig.2-79: Workspace, adding projects

To add a new project to the workspace, select in its context-sensitive menu: **Add ▸ New Project**.

Only one project can be opened in a workspace. If you open another project, the project opened before will be closed automatically.

Opening a Project

Open a project via its context-sensitive menu.

The status of the project is indicated by the icon in the tree structure.

Icon	Description
	Project available but not opened
	Project deleted or no connection to source computer
	Project opened in the project explorer

Fig.2-80: Project status, icons

Deleting a Project

A project can be removed from the workspace by using its context-sensitive menu.

☞ The project will not be deleted, only its reference is removed from the workspace.

2.4.5 Properties

The context-sensitive menu of a project can be used to display the properties of that project.

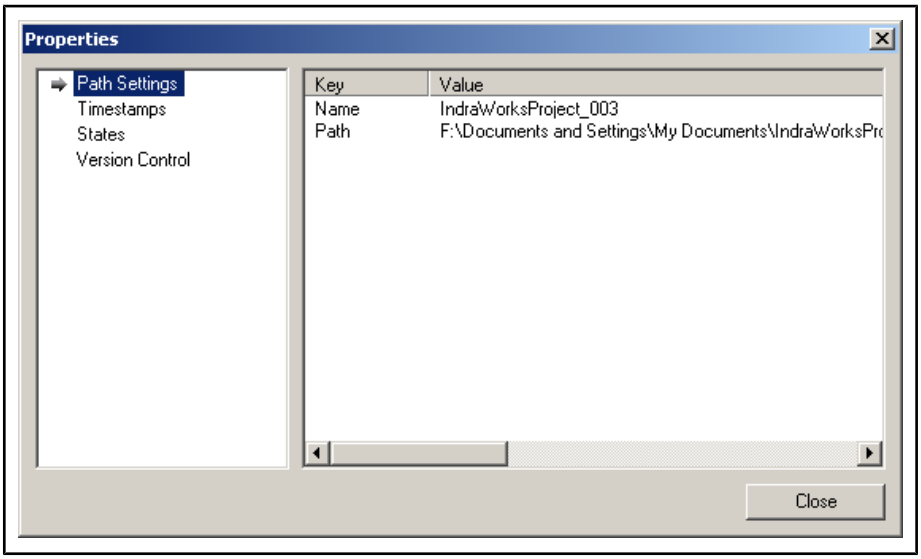


Fig.2-81: Properties dialog box

This dialog box displays the times when the project has been created and edited, the path and the state of the project as well as information on the project management in the version control.

The output can be specifically expanded by the devices inserted in the project.

2.4.6 Devices in Projects

“Top-level” devices are also displayed in the explorer of the workspace below their associated project. Use the context-sensitive menu to address functions for these devices and view properties of these devices. Devices which do not belong to the “top level” are not displayed in the explorer of the workspace.

2.5 Persistence

IndraWorks opens the previously opened project on the next start.

Windows arranged in the workspace before IndraWorks has been exited will be provided there after restart.

If the workspace is empty after start, you can create a new project or open an existing project.

2.6 Multilingual Projects

2.6.1 General Information

Creating and Editing Multilingual Projects

The language and text management integrated in IndraWorks allows to create and edit multilingual projects. The multilingual function relates to the language of the user texts (e. g. projectable messages, texts in HMI screens, program comments). Do not mistake this multilingual function for the language of the user interface.

We recommend to proceed as follows:

- Create the project and select the master language.
- Plan the project, all texts are entered in the master language.
- Add the desired additional languages to the project.
- Export the user texts of the project to one translation file per additional language.
- Translate the texts, i. e. edit the translation files.
- Import the translated texts to the project.

2.6.2 Defining the Master Language of a Project

The language used for creating an IndraWorks project is called the master language. Later, any number of further languages can be added to the project, in addition to the master language.

Define the master language in the “Project language” list box of the “Create New IndraWorks Project” dialog box while you create a new project.

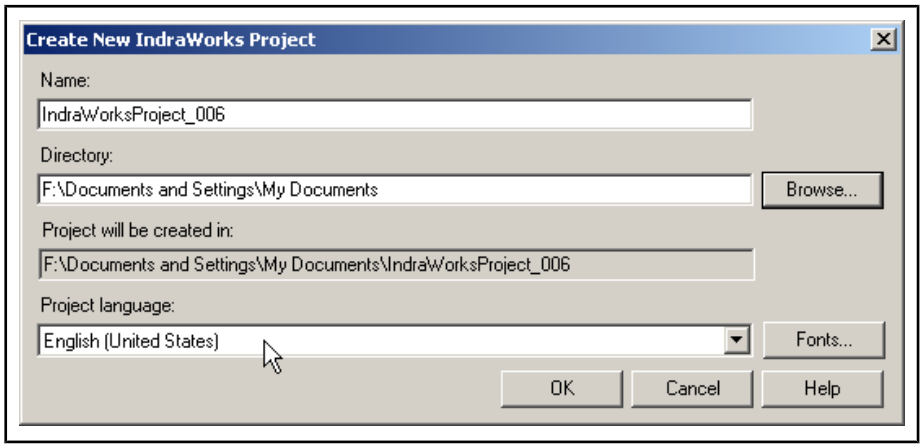


Fig.2-82: Create New IndraWorks Project dialog box, define master language

Use the “Fonts” button to define the fonts for the text display. This is particularly recommended for languages requiring specific fonts for displaying the text, e. g. Asiatic languages.

The default setting for the fonts of the master language is as follows:

Proportional font: Arial, 10 points

2.6.3 Managing Project Languages

Adding a New Language to the Project

To add a new language to the project, select **Project ▸ Language ▸ Manage Project Languages**.

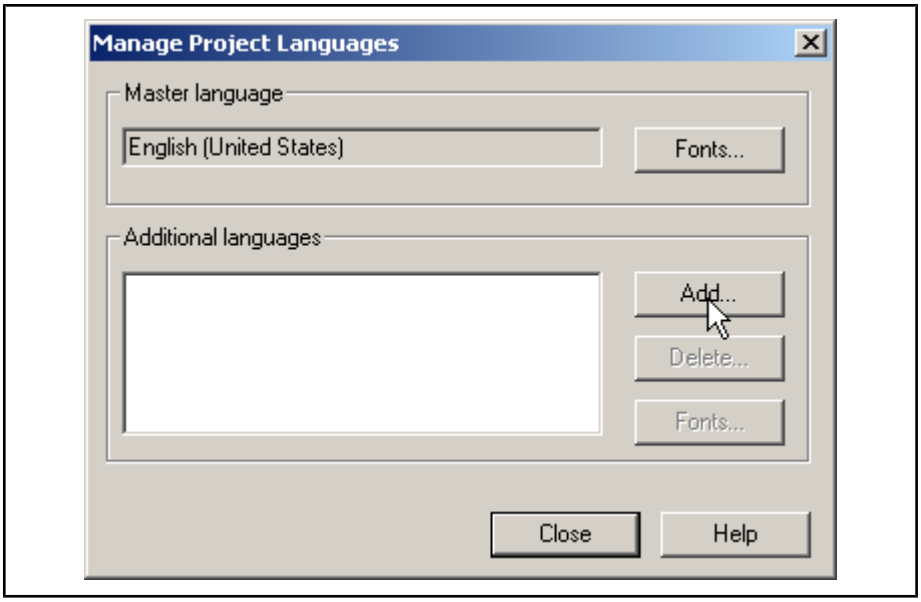


Fig.2-83: Manage Project Languages dialog box

To open the “Add Project Language” dialog box, click on “Add…” in the “Manage Project Languages” dialog box.

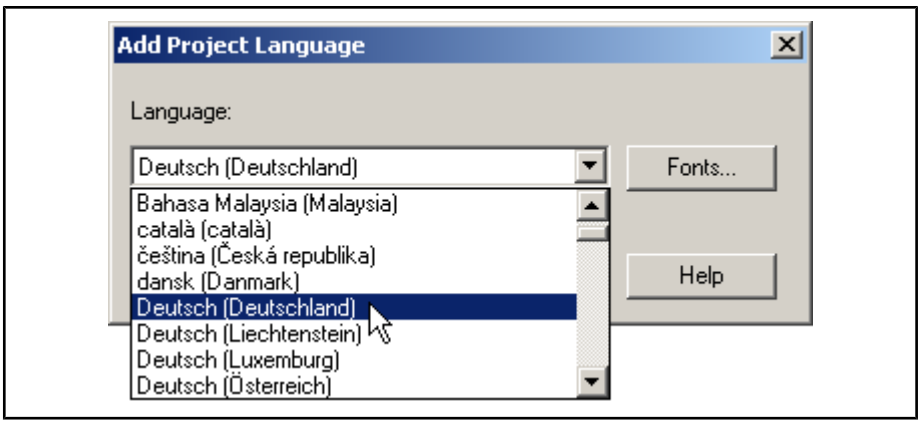


Fig.2-84: Add Project Language dialog box

Select the language you wish to add to the project.

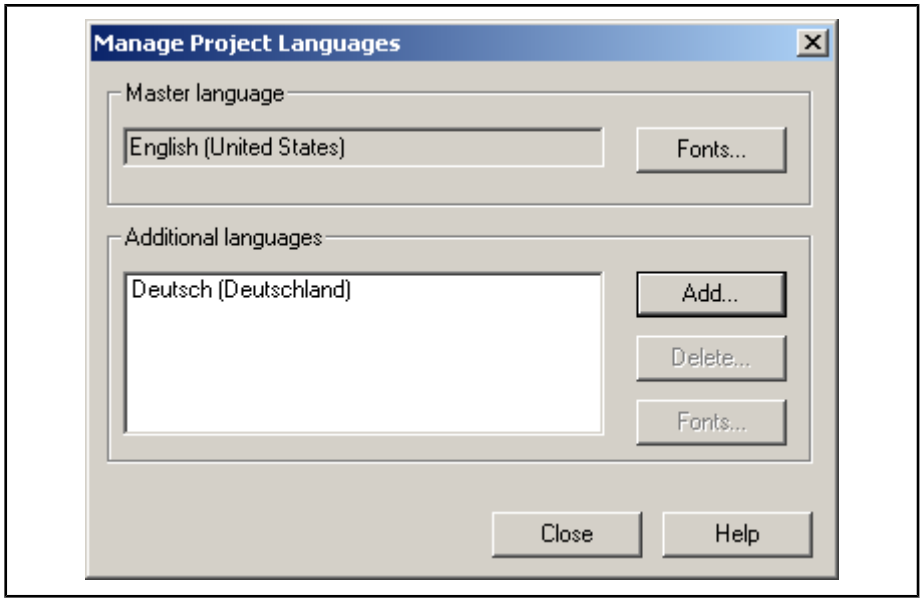


Fig.2-85: Manage Project Languages dialog box

Repeat this process until you have added all necessary languages to the project.

To change the font of an additional language, select the language from the “Additional languages” list and click on “Fonts…”.

Click on “Close” to return to the IndraWorks main window.

☞ When a new language is added, the internal text memories are prepared to incorporate and manage texts in this language. Translations to this language are, however, not made.

☞ Any added language can be selected as the active language. As a result, all language-specific user texts entered after this point are filed to the text memories of that language.

Editing in an additional language is only suitable for correcting a text or for making minor changes to a project.

For actual engineering, we explicitly recommend to use the master language only. Do not add other languages and do not translate user texts until the project has been completed technically.

If you wish to expand or revise a project, we also recommend to use the master language and translate the new user texts only thereafter.

Deleting an Additional Language from the Project

To delete a language from the project, select **Project ▸ Language ▸ Manage Project Languages**. Select the language to be deleted from the “Additional languages” list and click on “Delete…” Confirm the safety prompt.

2.6.4 External Project Translation

General Information

IndraWorks has got an export/import interface for the texts of a project. This data interface allows to export texts to a translation file which can then be translated outside of IndraWorks.

To do this, you need a spreadsheet program which is able to read and write *.csv files and which supports UNICODE.

For example, Microsoft Excel 2003 or OpenOffice.org Calc are suitable for editing translation files. The operation of these two programs differs only in the setting of text export and text import filters for opening and saving the translation files.

After the translation is completed, import the translation file. IndraWorks assigns the appropriate translations from the translation file to the original texts in the project.

The sections below describes the translation of the texts of a translation file by the example of OpenOffice.org Calc.

The following steps are required:

1. Export the translation file from IndraWorks.
2. Import the translation file to OpenOffice.org Calc.
3. Translate the texts. The translated texts are inserted in the translation file.
4. Export the translation file from OpenOffice.org Calc.
5. Import the translation file to IndraWorks.

Exporting the Translation File from IndraWorks

Open the “Export Translation File” dialog box via **Project ▸ Language ▸ Export Translation File**.

Fig.2-86: Export Translation File dialog box

Source Language

Select in this list box, which language version of the texts will be written to the translation file as original text.

The master language is preset in this field after opening the dialog box. However, you can also select any other language of the project as the source language.

Target Language

Select in this list box, which language will be the target language in the translation file. This entry will later be evaluated during import, so that the imported texts can be properly assigned to the correct text memories in the project.

Export Only Texts That are not Yet Translated

This check box allows you to define whether all texts are exported to the translation file or only those that do not have a translation in the target language yet. Check this option if you have revised an already translated project and wish to export the new texts only. When they will be imported, the texts of the translation file are merged with the already translated texts in the project.

Click on “OK” to open a dialog box where you can define the directory and the name of the translation file.

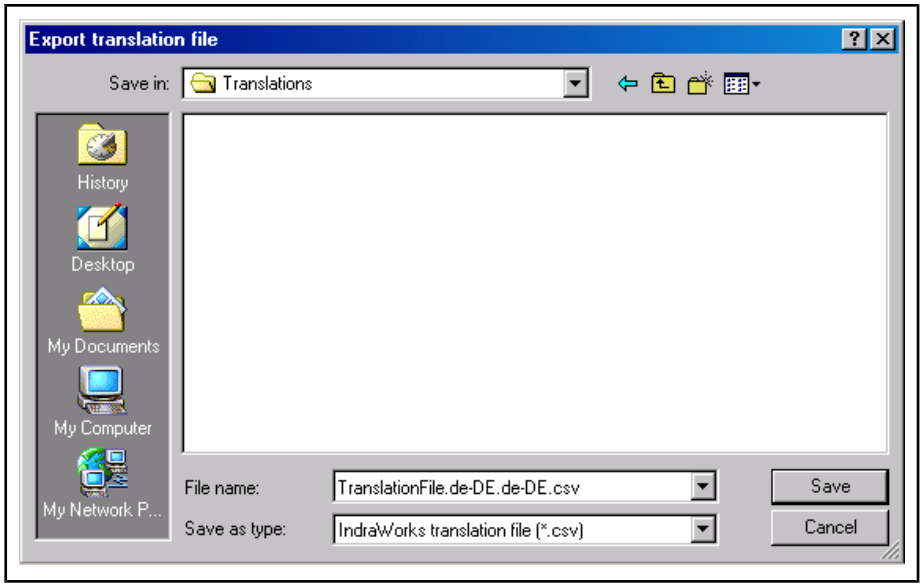


Fig.2-87: Export translation file dialog box, saving the file

The name proposed by IndraWorks has the following format:

`TranslationFile.SourceLanguage.TargetLanguage.csv`

For example, a translation file from German to US English:

`TranslationFile.de-DE.en-US.csv`

You can edit the name; but we recommend you not to change the name extension (e. g. `.de-DE.en-US.csv`).

Importing the Translation File to OpenOffice.org Calc

Download a translation file to OpenOffice.org Calc by **File ▸ Open**. Select the file to be opened and click on “Open”.

This opens the “Text Import” dialog box.

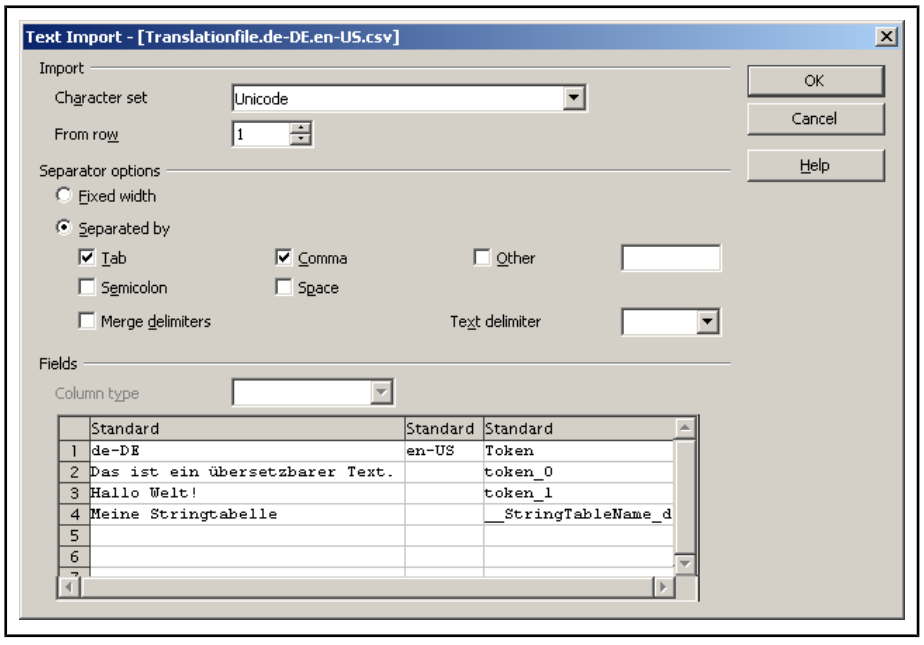


Fig.2-88: OpenOffice.org Calc: Text Import dialog box

Character Set “Unicode” must be set in this list box. This should already be preset because OpenOffice.org Calc automatically detects the character set used in the file.

Separator Options Select the separator option “Separated by” and “Tab” as separator.

Text Delimiter Text delimiters are not used in the translation file. Delete the suggested separator.

Click on “OK” to download the translation file to OpenOffice.org Calc.

Translating the Texts in OpenOffice.org Calc

After having downloaded the translation file to OpenOffice.org Calc, a spreadsheet is opened. It contains five columns and a number of lines depending on the number of texts.

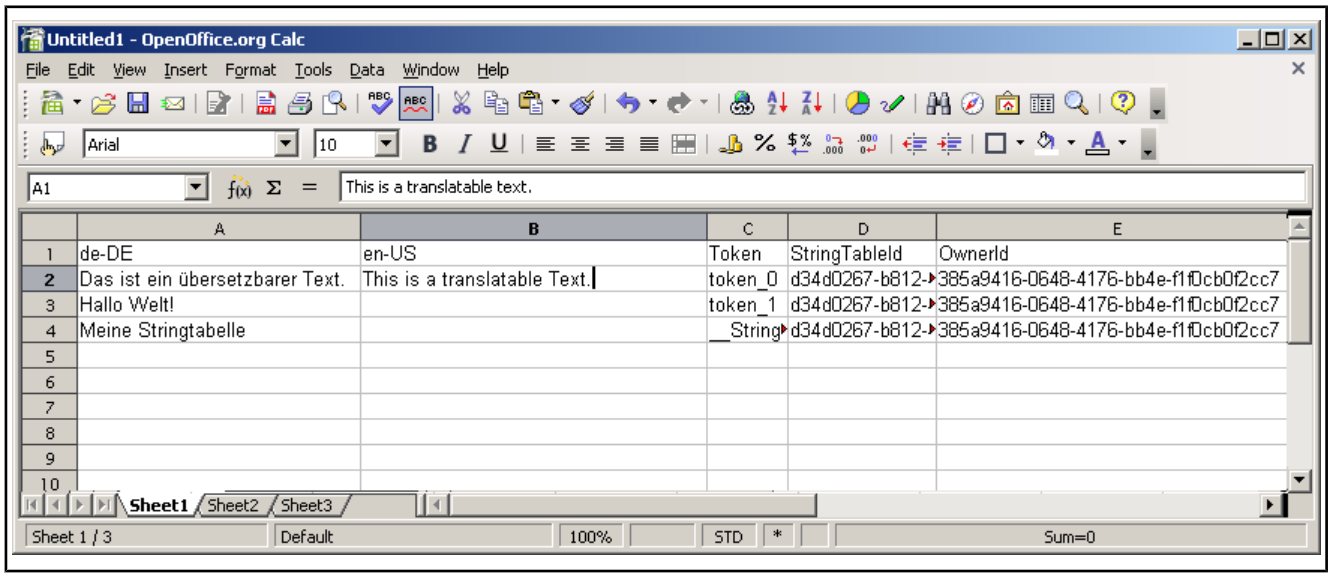


Fig.2-89: OpenOffice.org Calc: translating texts

The first line of the spreadsheet is a header which describes the column contents.

Column 1: Source Language Column 1 contains the texts of the source language. The header contains the ISO code for this language.

Column 2: Target Language Column 2 contains the texts of the target language. The header contains the ISO code for this language. Depending on the settings in the export dialog, the other lines of this column are empty or display already existing translations.

Columns 3, 4, 5: Management Data Columns 3 to 5 contain management data which will be used to assign the translated texts properly when they are imported to IndraWorks.

Do not change the header or the management data. Otherwise, the translation file cannot be imported.

During the translation, the translated texts are entered in column 2. While being imported, these texts are read in IndraWorks.

Please note the following character strings for the translation: “{%tab%}”, “{%lf%}”, “{%crlf%}”

During the export, these character strings are inserted in texts which contain tabulator and end-of-line characters. Do not change these character strings in the translation.

Example:

Source text: Hallo{%tab%}Welt!!!

Translation: Hello{%tab%}World!!!

When the translation file is imported to IndraWorks, the character strings are reconverted into the original characters.

Exporting the Translation File from OpenOffice.org Calc

After the translation is completed, save the translation file. We recommend to save the edited file under a different name with **Datei ▸ Speichern unter** (File Save As).

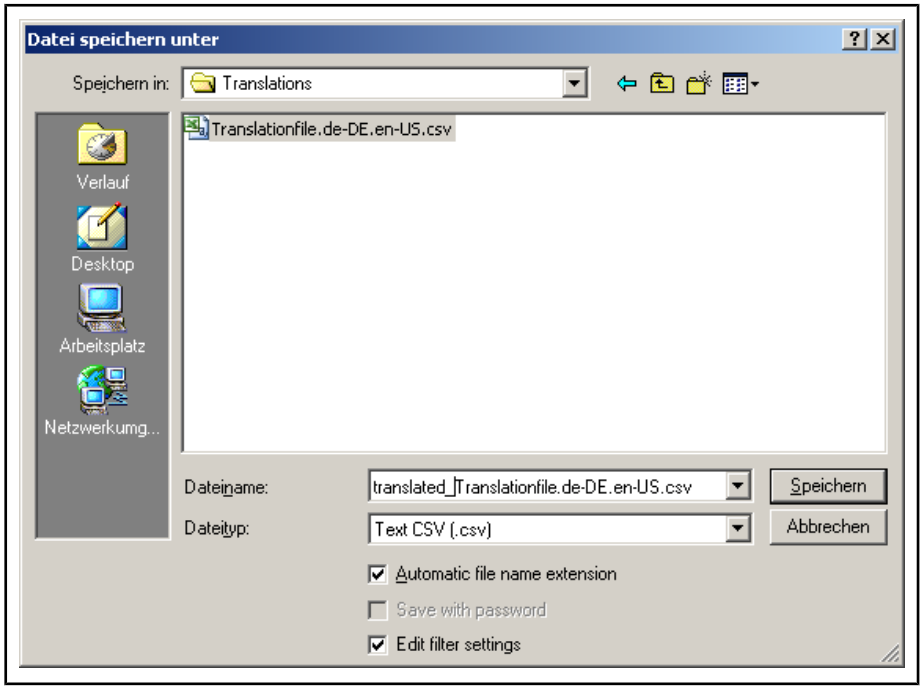


Fig.2-90: OpenOffice.org Calc: Datei Speichern unter (Save File As) dialog box, exporting the translation file

Dateityp (File type)

Select “Text CSV” from the “Dateityp” (File type) list box.

Editing Filter Settings

The “Edit filter settings” check box allows you to select the filter settings before saving the file in the “Export of text files” dialog box.

To open the “Export of text files” dialog box, click on “Speichern” (Save).

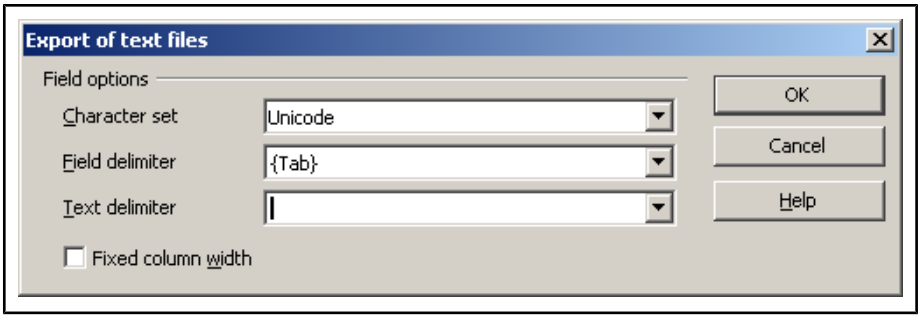


Fig.2-91: OpenOffice.org Calc: Export of text files dialog box

Character Set Select “Unicode” from this list box.

Field Delimiter Select “{Tab}” as “Field Delimiter”.

Text Delimiter Delete the character in the “Text delimiter” field.

The export filter must be set exactly as described above because, otherwise, the translation file cannot be imported to IndraWorks.

Importing the Translation File in IndraWorks

To import the texts of a translation file to the current project, select **Project ▸ Language ▸ Import translation file**. From the file selection dialog box, select the translation file to be imported.

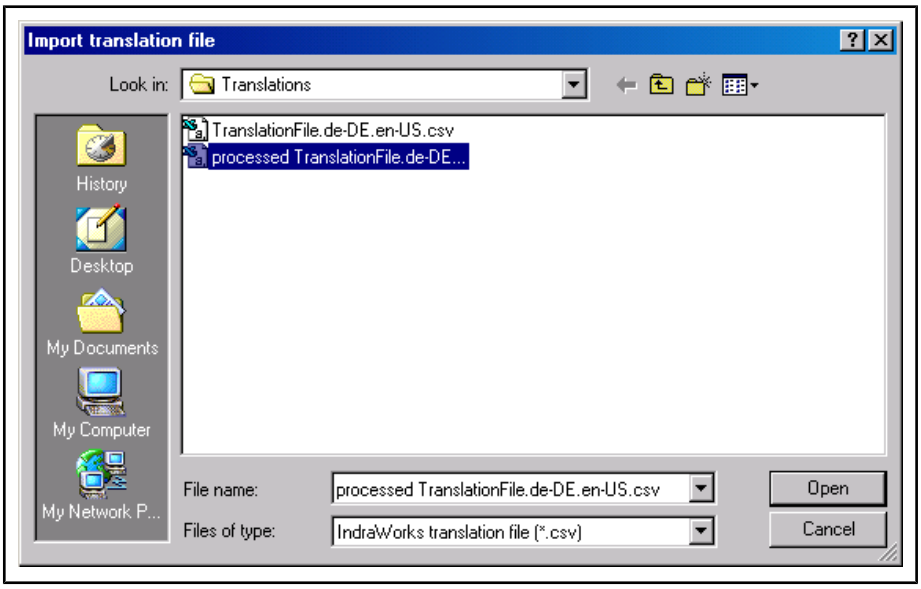


Fig.2-92: Import translation file dialog box, selecting the file

You can now set the import options in the “Import translation file” dialog box.

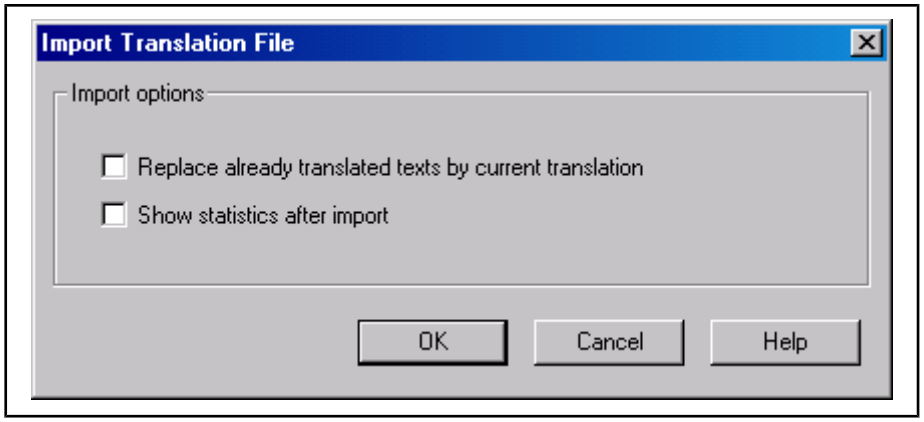


Fig.2-93: Import translation file dialog box

Replace Already Translated Texts by Current Translation

Activate this check box to replace already existing translations by the current translations. For example, this is reasonable if the translation file contains a complete set of texts which have been revised in their language and are now to be applied to the project.

If you do not activate this check box, any already translated texts in the project will not be changed; the corresponding texts of the translation file will be ignored.

Show Statistics After Import

Activate this check box to view import statistics after the import.

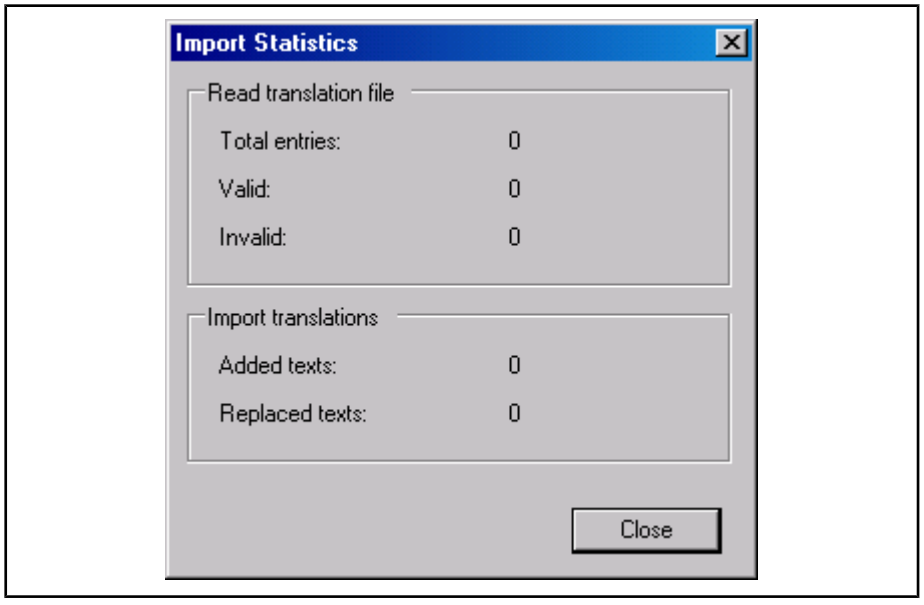


Fig.2-94: Import Statistics

Total Entries

Total number of lines read from the translation file.

Valid

Number of the lines in the translation files that have been rated as valid.

Invalid

Number of the lines in the translation files that have been rated as invalid. Invalid lines develop mainly if the export filter has not been set properly on translating and saving the translation file in the external program.

Added Texts

Number of new texts imported to the project.

Replaced Texts

Number of the texts already translated in the project which have been overwritten by the imported current text.

2.7 Printing

2.7.1 General Information

This IndraWorks version provides the functionalities Print, Print Preview and Configuration of Print Settings to document your data.

You can print

- Device Data
- Project data

It depends on your current working context, which menu entries for printing are available. The entries can be selected only if a printable element (area or node) has been selected.

2.7.2 Print Settings

General

Use **File ▸ Print Settings...** or **Print Settings...** in the context-sensitive menu of the project node to configure the printing in IndraWorks. You can configure the layout of the printed document as well as the header and footer. The settings apply for the current project. They can be used for other projects via the export/ import functionality.

Basic Buttons

OK Use “OK” to save all changes and to close the dialog box.

Cancel Use “Cancel” to discard all changes including those resulting from import and to close the dialog box.

Default Settings Use “Default Settings” to reset all settings to the default values.

Page Settings Use “Page Settings” to open the dialog box “Page Settings”. Here you can define your settings for borders, orientation, paper size and you can select a printer.

Export The menu entry “Export” enables you to export the print settings of the current project and to apply them to another project. Use the button “Export” to open the dialog box “Export Print Settings”. Here you can enter a name and the destination directory for the export file, confirm by “OK”.

Import The menu entry “Import”enables you to import the print settings exported before. Use the button “Import” and select in the appearing dialog box “Import Print Settings” a file exported before. Confirm by “OK”. Thus the import will be done and the imported print settings will be displayed.

Help Use “Help” to start the IndraWorks online help.

Table of Contents

On the tab “Table of Contents” you can configure the layout of the table of contents and the cover page. These pages will be created automatically when project data or device data are printed.

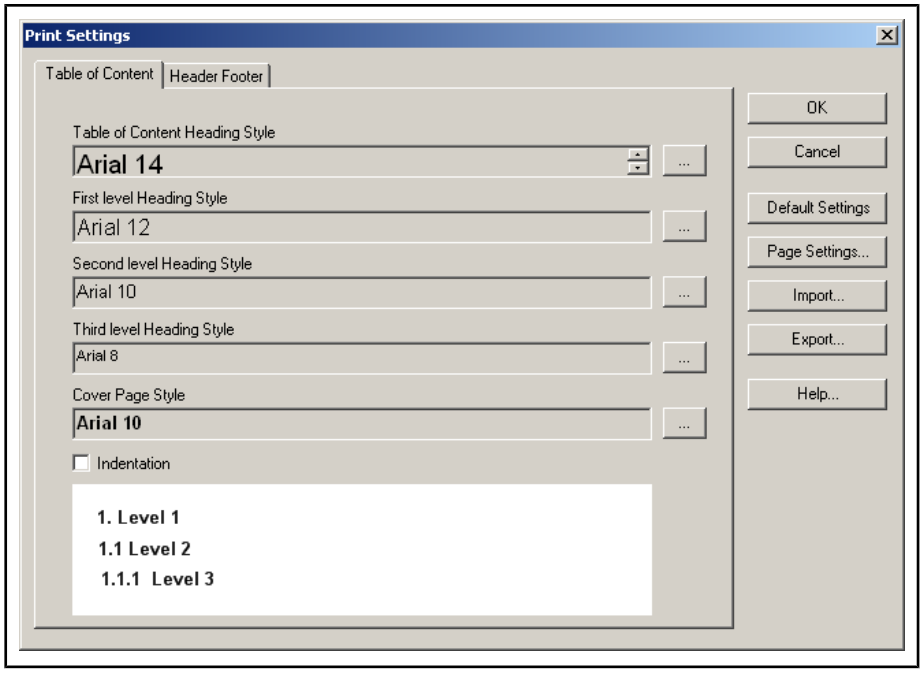


Fig.2-95: Print Settings dialog box, layout of table of contents and of cover page

In order to define the font for the cover page or the fonts for the heading levels in the table of contents press the button "..." on the right of the according input box. Select the desired font in the Font dialog.

Use the checkbox "Indentation" to display the table of contents indented or left-aligned.

Header, Footer

The tab "Header Footer" contains a print preview of header and footer. You can configure those via the "Edit ..." button.

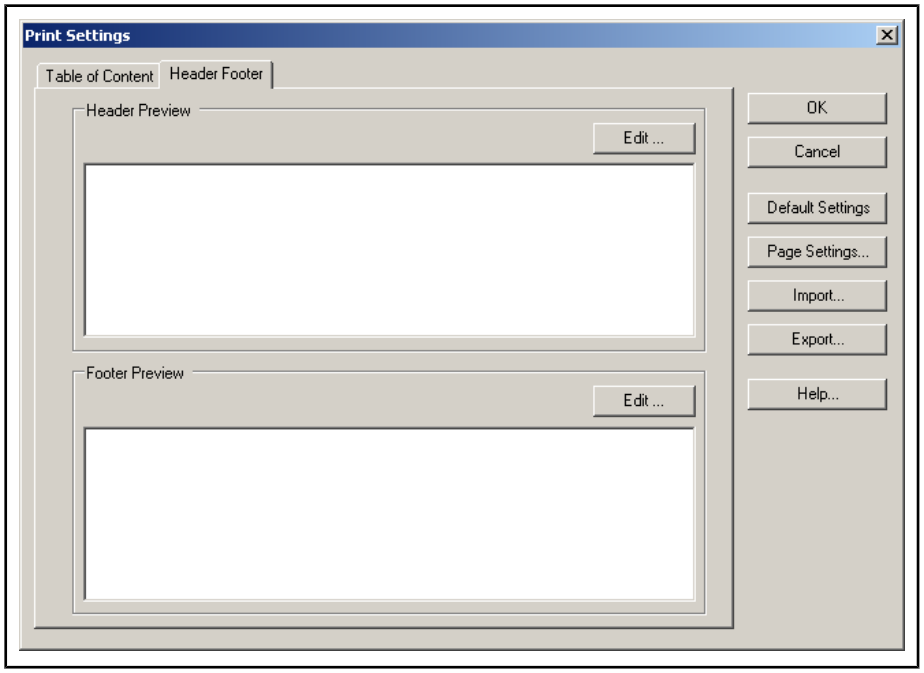


Fig.2-96: Print Settings dialog box, header and footer

Open the edit dialog for header and footer by the “Edit ...” button. It will be described how to edit the header in the following section. The footer can be edited in the same way.

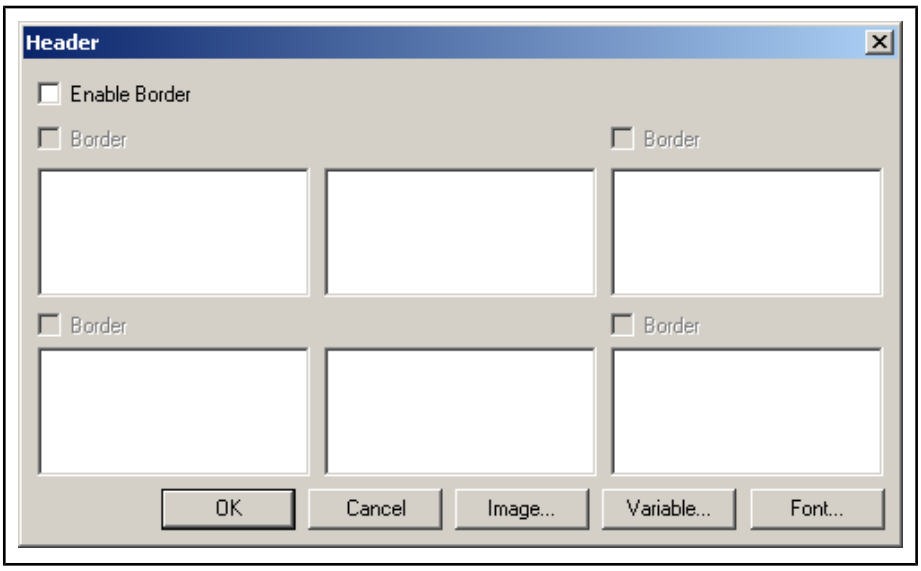


Fig.2-97: Print settings, edit header

This dialog box contains six sections, which can be filled with text or graphics. The three upper sections are used to configure the upper part of the header, the three lower sections are used for the lower part. Activate the “Header Border” checkbox to separate these two parts of the header by a line. Then you can show separation lines between the various sections by the “Border” checkboxes.

The font can be separately set for each section by using the “Font...” button. You can add variables, such as page number or date, via the “Variable...” button. Use the “Image...” button to add graphics.

☞ If you insert a graphic into a section, you cannot insert additional text. Graphics cannot be displayed together with text in the same section.

Switch back to the print preview for header and footer by “OK” and check your settings. Header and footer will not be printed, unless data have been entered in the edit dialog box.

2.7.3 Printing Project Data or Device Data

Overview

To print IndraWorks project data select **Print** in the context-sensitive menu of the project node or in **File** of the main menu.

☞ Independent of the project node you can start the printing by **Project ▸ Printing Global Documentation...**.

To print data of single devices use **Print** in the context-sensitive menu of the device node or in the main menu under the entry of the selected device.

The menu entries **Print** and **Print Preview** provide the following pages:

Cover Page The cover page contains all project related information: project name, project description, date of creation and date of last modification.

You can change the font of the cover page via **File ▸ Print Settings**.

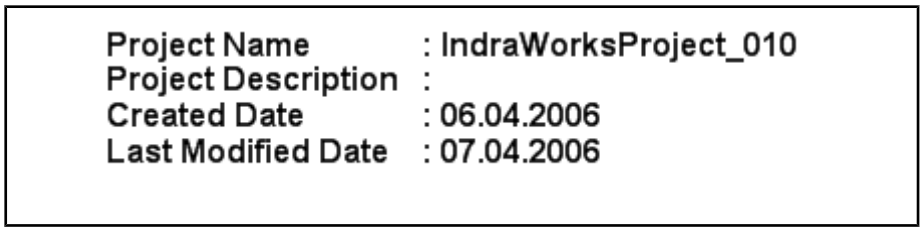


Fig.2-98: Printing, print preview of projects - cover page

Table of Contents

This page contains the following information:

1. The first heading level displays the device name.

1.1. The second heading level displays the data categories of the device.

1.1.1. The third heading level displays the file name of the printed data category.

You can change the fonts of the table of contents via **File ▸ Print Settings**.

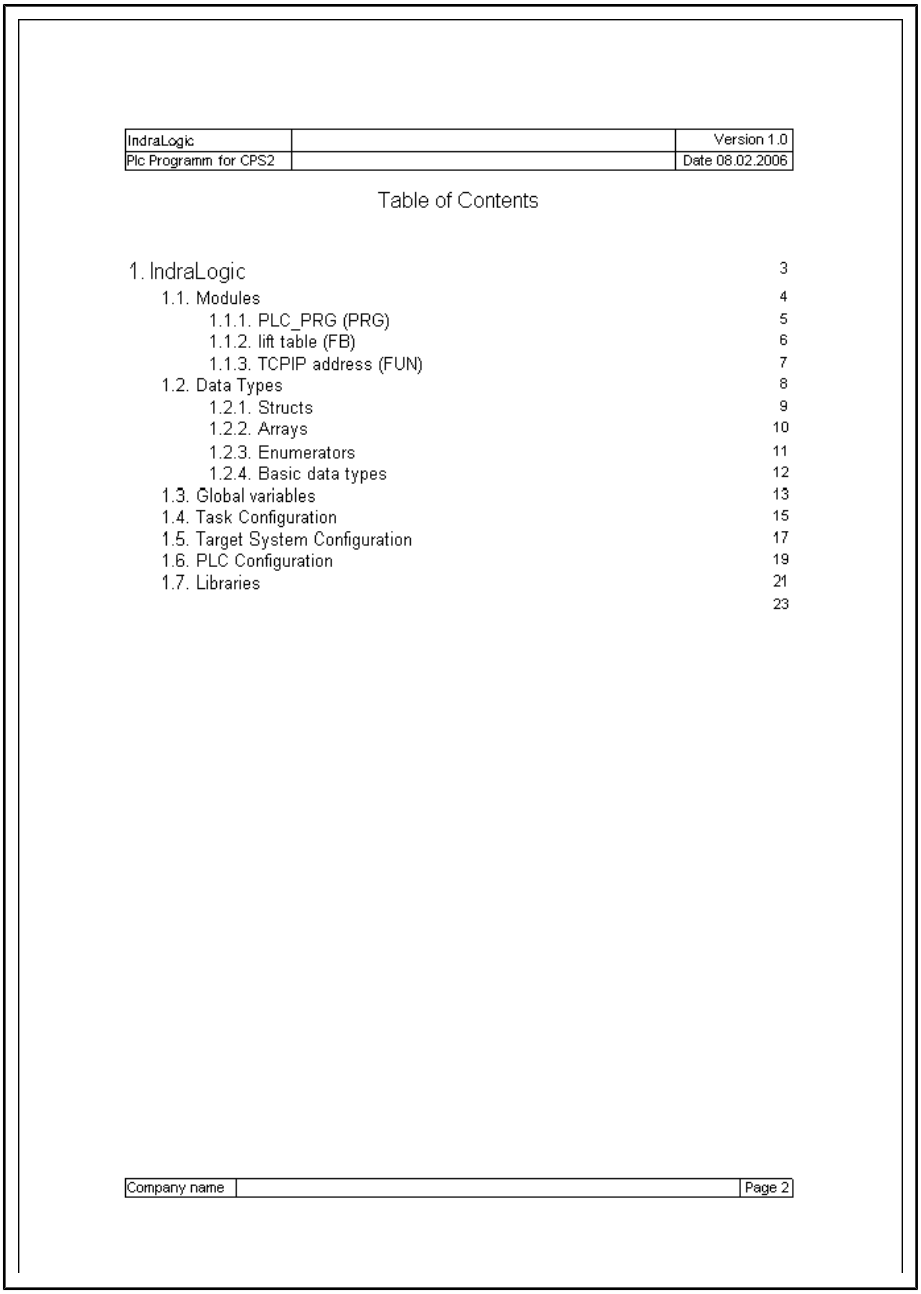


Fig.2-99: Printing, print preview of projects - table of contents

Separator Sheet - Device

This page contains the name and the description (optional) of the selected device.

Separator sheet and cover page have the same font. You can change this font via **File ▸ Print Settings**.

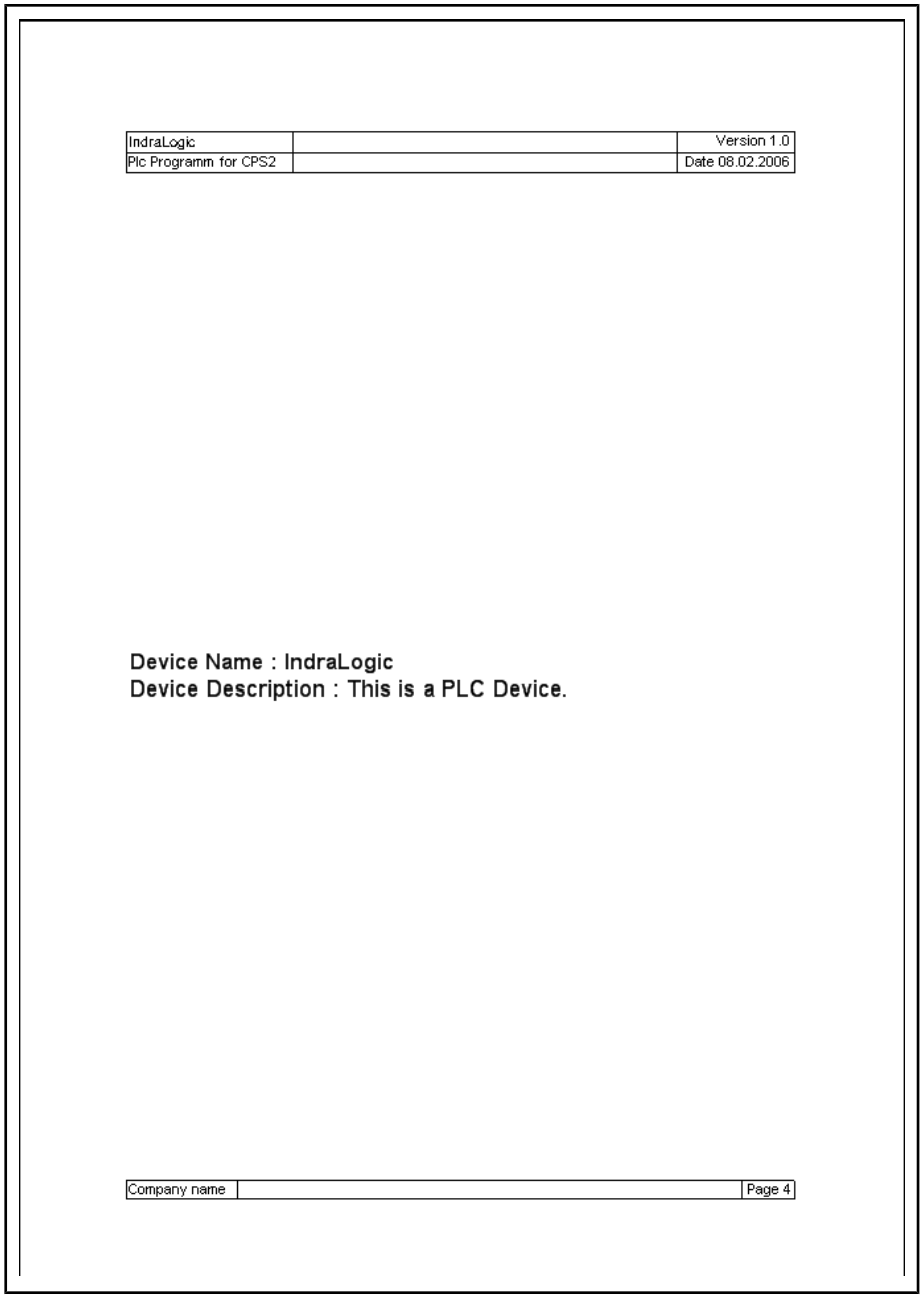


Fig.2-100: Printing, print preview of projects - separator sheet - device

Separator Sheet - Device Data

This page contains the name of the device data.

Separator sheet and cover page have the same font. You can change this font via **File ▸ Print Settings**.

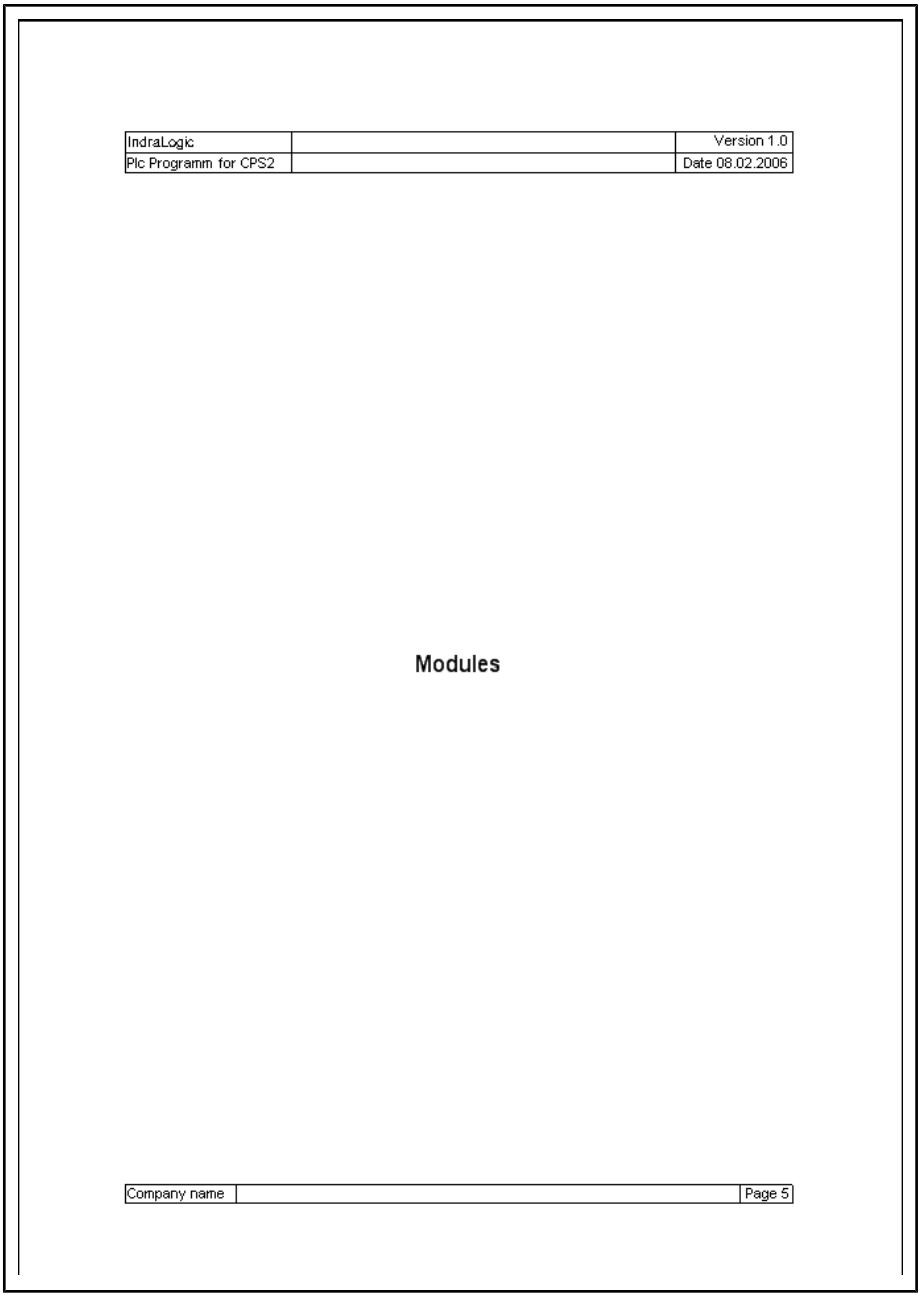


Fig.2-101: Printing, print preview of projects - separator sheet - device data

Device Data This page contains device data and device specific information.

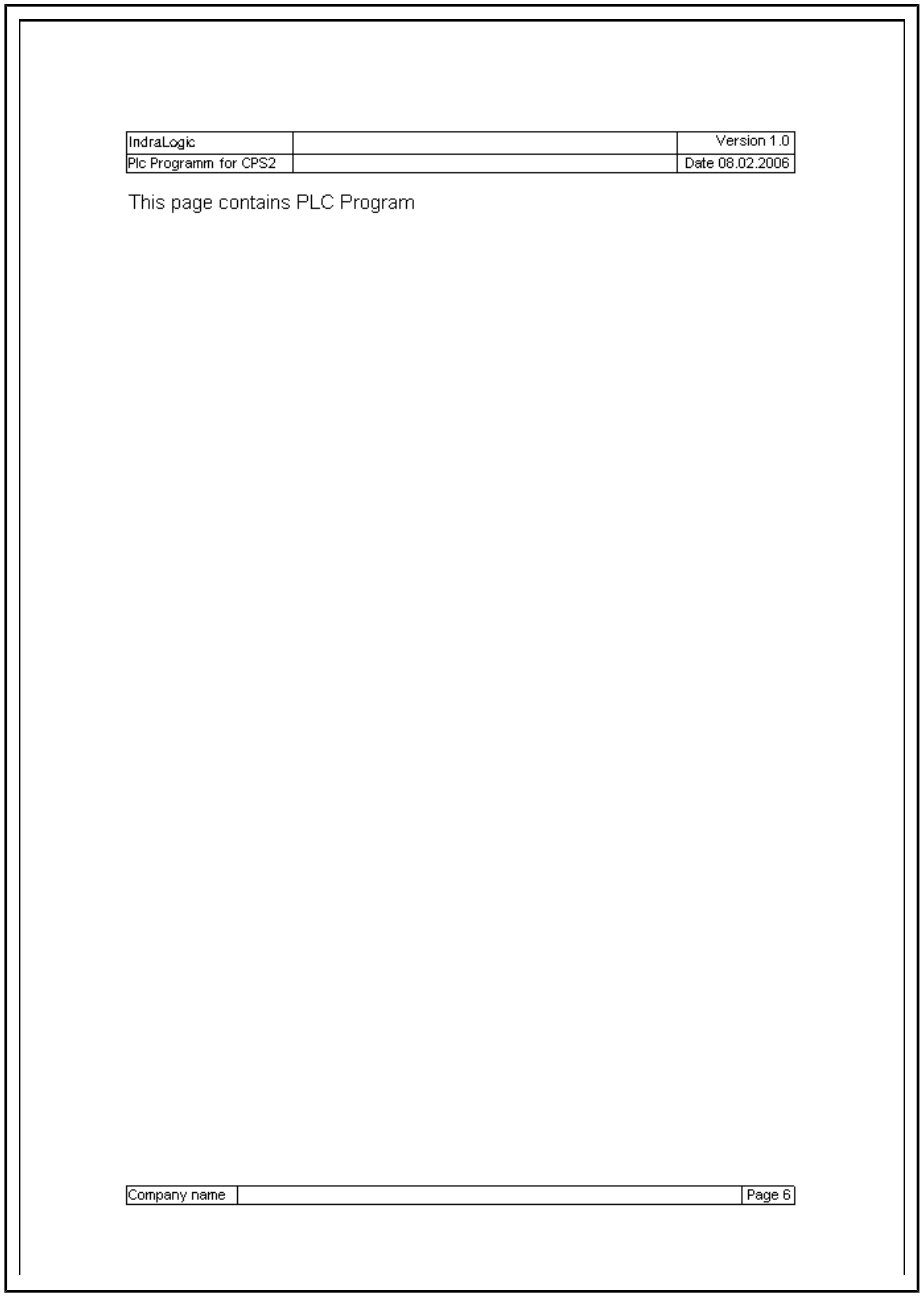


Fig.2-102: Printing, print preview of projects - device data

Print and Print Preview - Operating Mode

Printing and Print Preview of Project Data

To print or preview IndraWorks project data proceed as follows:

Step 1 Select a project with printable devices.

Step 2 Open the context-sensitive menu of the project node. Select **Print** or **Print Preview**. Or select **File Print** or **Print Preview** in the main menu, if the project node is selected.

Independent of the project node you can select **Project ▸ Printing Global Documentation...** or **Project ▸ Print Preview Global Documentation...**.

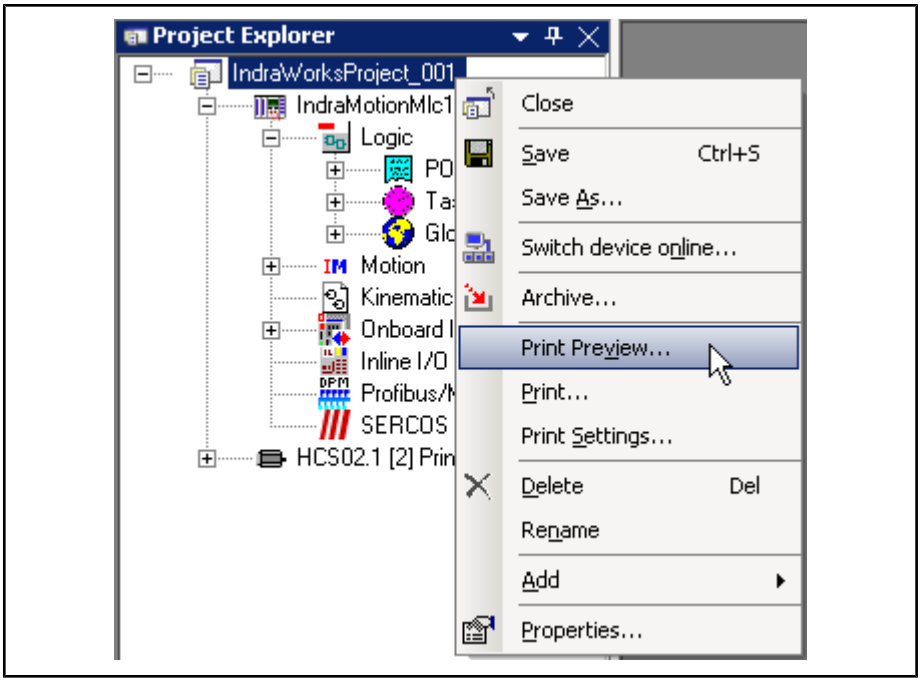


Fig.2-103: Print, print preview of projects – select command

Step 3: Selecting the Printable Data

The following dialog box displays all printable devices and nodes of the project. If you select a device or printable node in this tree structure, the according data categories will be displayed in the right list, if available. The data categories display the printable data of the selected node. Select the devices and nodes and their data categories to be printed.

If you select a node in the project tree, then you can use its context-sensitive menu to select or deselect all printable data of this node and of its attached nodes.

Press “Next>>”.

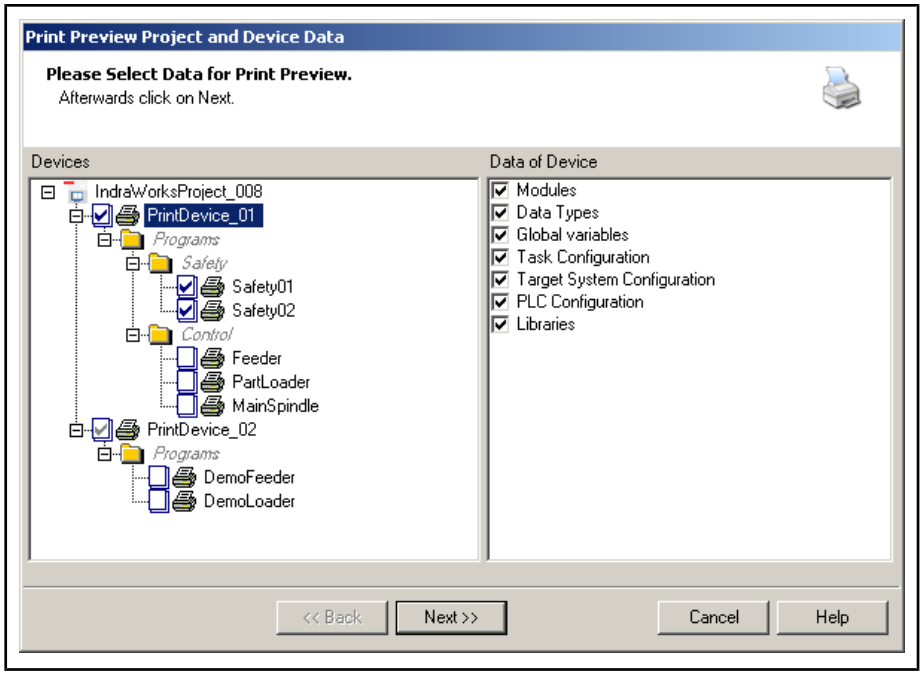


Fig.2-104: Print Project and Device Data dialog box – device selection

Step 4: Checking

You can check your settings in the next dialog box.

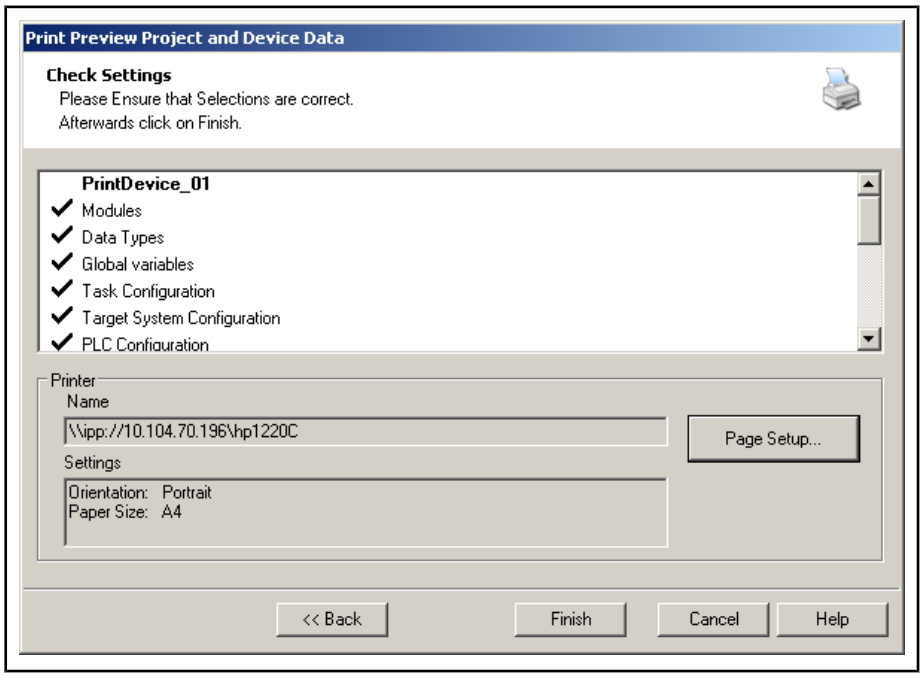


Fig.2-105: Print Project and Device Data dialog box – check settings

1. The dialog box displays the selected devices and nodes and their data categories with a check mark.
2. The “Name” field in the “Printer” section displays the currently selected printer. If necessary, select a different printer via “Page Setup”.
3. The “Settings” field contains the values for page orientation and paper size.

- If you want to change the settings for margins, orientation, paper size or the printer use the “PageSetup” button. The new settings will be displayed in the “Printer” section and are persisten for the IndraWorks session.

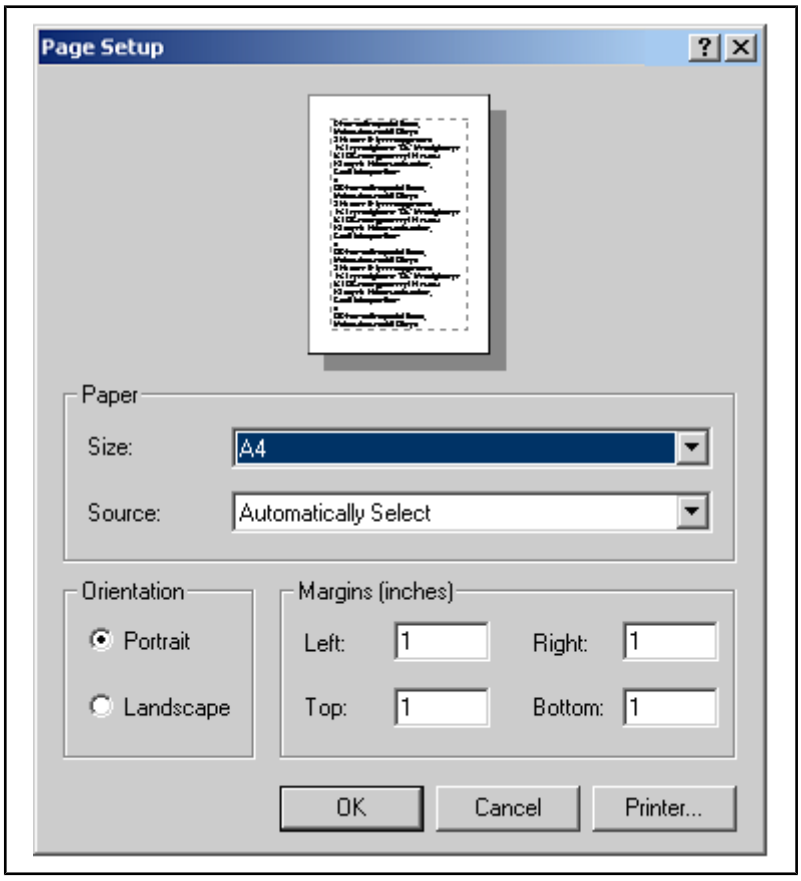


Fig.2-106: Page Setup dialog box

4. Click on “Finish” to start the printing or preview.

Step 5: Progress Bar A progress bar will be dispalyed till all the pages have been prepared for printing or preview.

You can stop this process by “Stop”. However, the already prepared pages will be printed or previewed.

Printing and Print Preview of Device Data

To print or preview device data proceed as follows:

Step 1 Select a project with printable devices.

Step 2 Open the context-sensitive menu of the device node. Select **Print** or **Print Preview**.

Step 3: Device Selection The following dialog box displays the selected device and all attached printable nodes of this device. If you select the device or a printable node in this tree structure, the according data categories will be displayed in the right list, if available. The data categories display the printable data of the selected node. Select the nodes and their data categories to be printed.

☞ If you select a node in the project tree, then you can use its context-sensitive menu to select or deselect all printable data of this node and of its attached nodes.

Press “Next>>”.

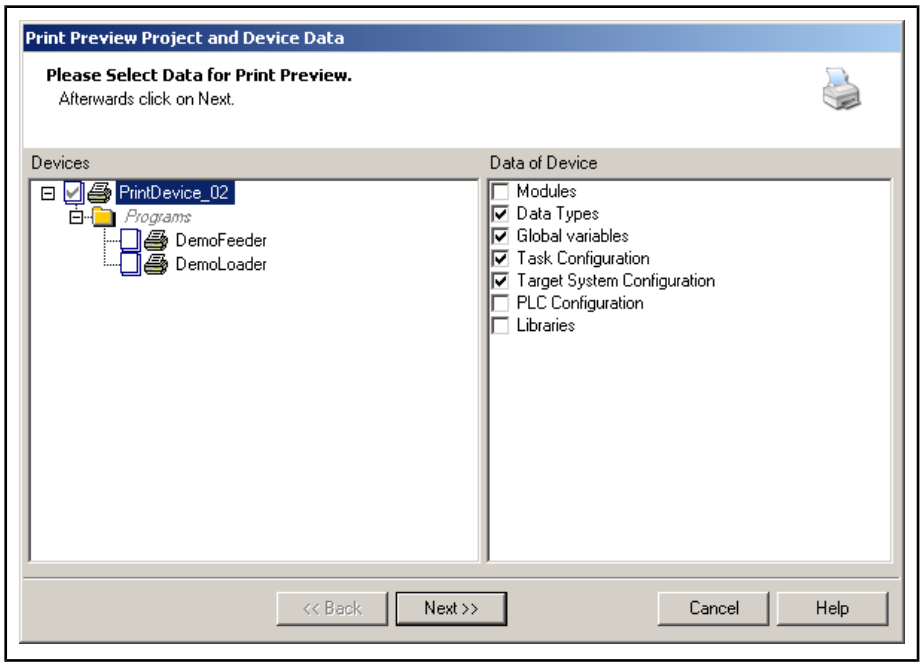


Fig.2-107: Print Project and Device Data dialog box, select device data

Step 4: Checking

You can check your settings in the next dialog box.

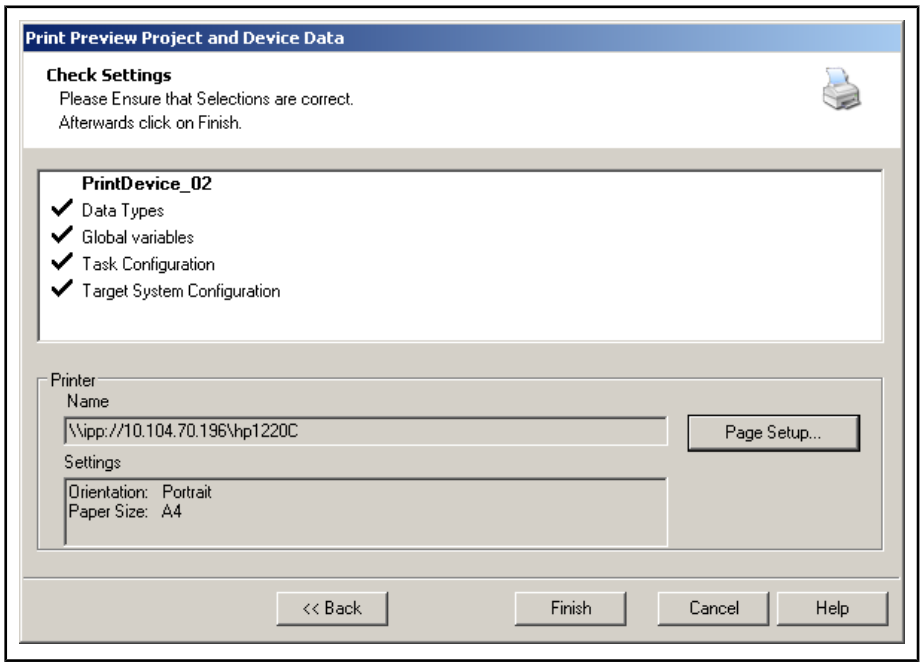


Fig.2-108: Print Project and Device Data dialog box, check settings

1. The dialog displays the selected data of the device.
2. The “Name” field in the “Printer” section displays the currently selected printer. If necessary, select a different printer via “Page Setup”.
3. The “Settings” filed contains the values for page orientation and paper size.
   - If you want to change the settings for margins, orientation, paper size or the printer use the “PageSetup” button. The new settings will be

displayed in the “Printer” section and are persisten for the IndraWorks session.

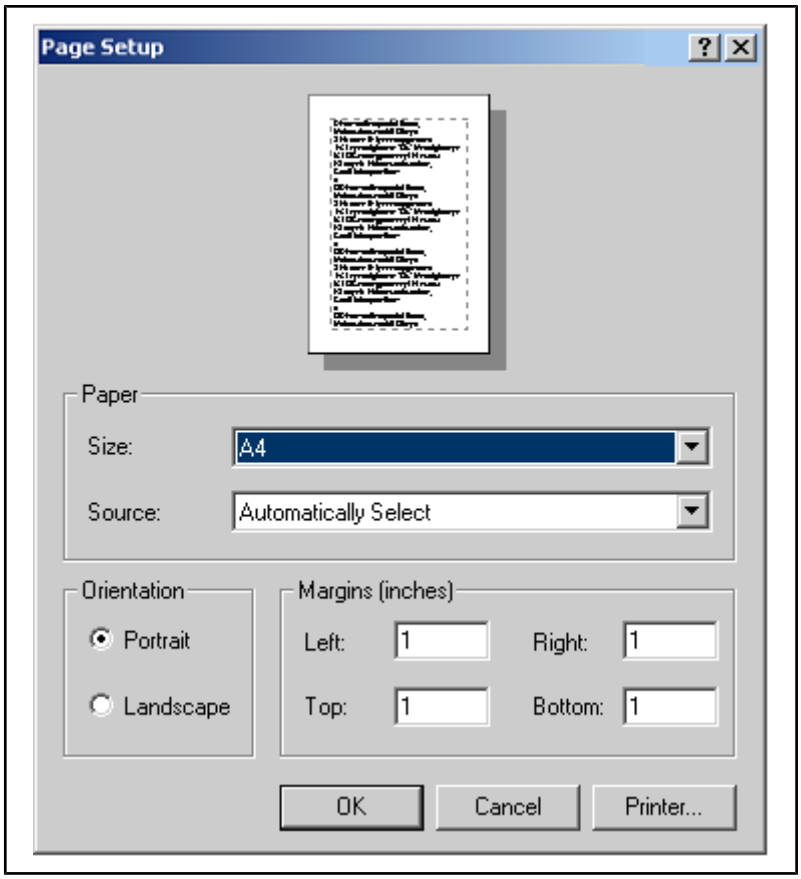


Fig.2-109: Page Setup dialog box

4. Click on Finish to start the printing or preview.

2.8 User Management

2.8.1 General Information

Functioning of the User Management

The user management makes sure, that only authorized persons can access protected functionalities in IndraWorks. To access such a protected functionality, a user has to login in the user management with his user name and password. After that, the privilege system of the user management checks the permissions of the user and declines to execute the function unless the user has got the necessary permissions.

User Management in the IndraWorks Project

User management data are components of an IndraWorks project. They will be loaded when the project is opened. Different IndraWorks projects have got different user data bases. The user management will not become active until an IndraWorks poject has been opened, since the information on users or permissions will not be available before.

When a new IndraWorks project is created or when a project created by an older IndraWorks version is opened, an initial user data base with exactly one user will be created. This user is the administrator (user name “Admin”). On principle, the administrator has got all permissions and thus can access all protected functions. The user account of the administrator should be used only for administrative tasks, e. g. to configure the user data base (create users and groups, assign permissions, etc.).

Users, Groups, Permissions

A user represents a person, which has to work with IndraWorks and thus needs certain permissions. Users are identified by a unique user name. They possess further properties, such as first name, last name, description, settings, etc. Users in IndraWorks do not get permissions directly, but they get the permissions of the group they are members of.

Groups allow to define permission profiles and to assign them to users. Permissions are assigned to groups, and users are added to groups. A user possesses all permissions of all groups he is added to.

A permission allows to access a certain protected functionality in IndraWorks.

Administrator

The administrator (user name “Admin”) is the super user in IndraWorks. The administrator already exists in an initialized user data base, immediately after creating a new IndraWorks project. He possesses all permissions, without any exceptions. The user data of the administrator cannot be changed, except some settings. The administrator cannot be deleted.

Default Groups

The user management provides predefined default groups with characteristical permission profiles of frequently found user groups. This simplifies the configuration and the assignment of permissions. Add users to the default groups, in order to assign typical permission profiles. Thus the configuration effort will be minimalized.

Default groups cannot be edited or deleted. But you can copy a default group and modify this copy as you want. The copy of a default group is a “normal” group, i. e. it can be edited and deleted.

Initial Password

If you create a new user, it will get an empty password. To login a user the first time, only the user name must be entered. The user management recognizes this first login and asks the user to define a password.

2.8.2 Activating the User Management

The user management is disabled when you create a new IndraWorks poject or open a project created by a former IndraWorks version. You can run all IndraWorks functions without being logged in, even protected functions.

To monitor protected functionalities, you have to activate the user management. For this purpose select **Project ▸ User Management ▸ User Management Active**.

Once you have enabled this option, only privileged users can run protected functions. A login will be necessary.

To disable the user management deselect **Project ▸ User Management ▸ User Management Active**. However, unless you are logged in as administrator you cannot disable the user management .

2.8.3 Login and Logout, Change a Password

First Login as Administrator

If you activate the user management in an IndraWorks project for the first time, an initial user data base is created with just one user, the administrator. This data base is a component of the IndraWorks project data.

To login as administrator in IndraWorks proceed as follows:

1. Select **Project ▸ User Management ▸ Login User**
2. Enter only the user name “Admin” in the login dialog, enter no password. Confirm with “OK”.

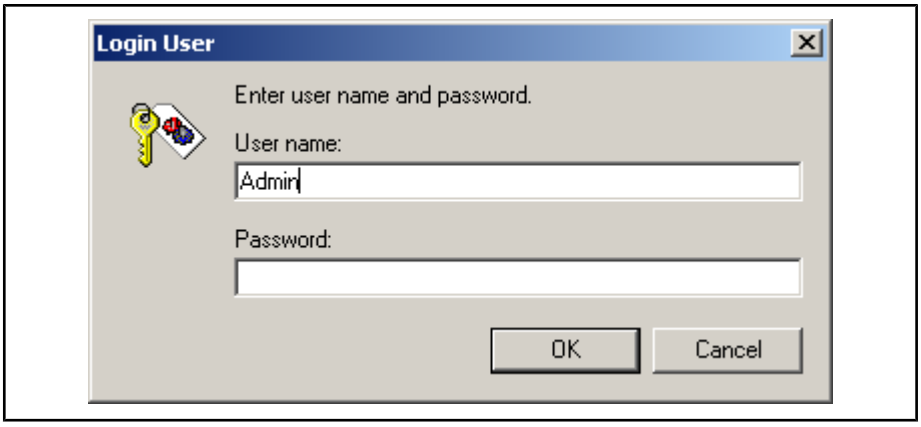


Fig.2-110: Login User dialog box, first login of "Admin"

3. The user management recognizes this first login as administrator and asks you to enter an administrator password. Confirm the following message by “OK”.

Fig.2-111: User Management, recognizes first login

4. Enter and confirm the new password for the administrator now. Finish the entry by “OK”. From now on you will have to enter the user name “Admin” and the defined password, if you login as administrator.

Fig.2-112: Enter Password dialog box, enter and confirm new password

You are logged in as the administrator in IndraWorks now and can configure the user management, i. e., create the necessary users and groups and assign permissions. To logout select **Project ▸ User Management ▸ Logout User**

User Login

1. To login in IndraWorks select **Project ▸ User Management ▸ Login User**
2. Enter your user name and password in the login dialog and click on “OK”.

If the user name and the password are correct, you will be logged in. Otherwise you will get an error message.

If you log in for the first time in IndraWorks or if your password has been reset by the administrator, IndraWorks will ask you to enter a new password.

User Logout

1. To logout in IndraWorks select **Project ▸ User Management ▸ Logout User**
2. A message confirms the logout.

Change Password dialog box

1. Login in Indra Works first, to change your password.
2. Select **Project ▸ User Management ▸ Change Password**.
3. In the “Change Password” dialog box, enter the old and the new password, confirm the new one and click on “OK”.

Fig.2-113: Change Password dialog box

Login with Reference Code and Key Code

The login with reference code and key code allows a login in IndraWorks with administrator permissions. This login is meant for emergencies, e. g., if the administrator cannot remember his password or if no login is possible for other reasons. To login with reference code and key code proceed as follows:

1. Select **Project ▸ User Management ▸ Login with Reference Code and Key Code**.
2. The “RefCode” field of the login dialog displays an eight-digit reference code. Send this reference code to the Bosch Rexroth service. Do not close the login dialog.

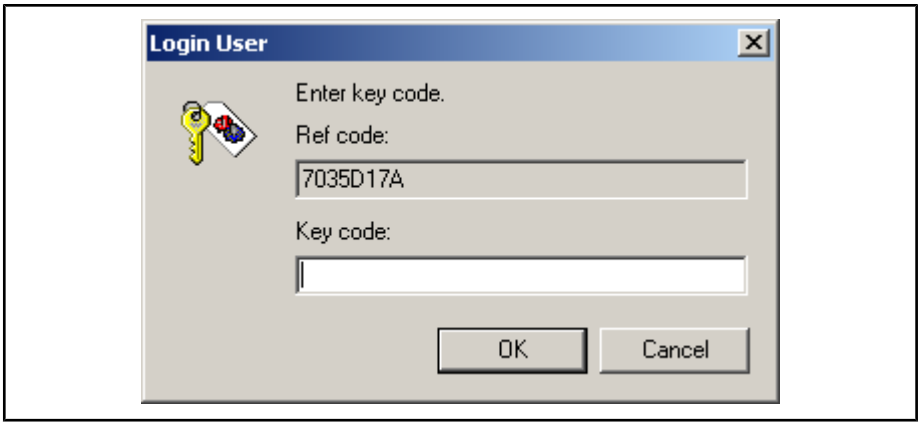


Fig.2-114: Login User dialog box, with reference code and key code

3. You will get a key code belonging to your reference code from the Bosch Rexroth service. Enter this code in the “KeyCode” input box of the login dialog and click on “OK”. Now you are logged in as administrator.

A key code works only together with the according reference code. Each login provides a new reference code. You cannot use old key codes.

All functions can be executed (administrator permissions) after the login with reference code and key code. Please, be very careful and logout when you have finished working in IndraWorks.

2.8.4 Configuring the User Management

User List and Group List

To display the user list or the group list of an IndraWorks project open this project and select **Project ▸ User Management ▸ Configure**. The “Configure User Management” dialog box will be opened. This dialog box consists of two tab pages, one containing the user list, the other one containing the group list. Click on the tabs “Users” or “Groups” to display the according list.

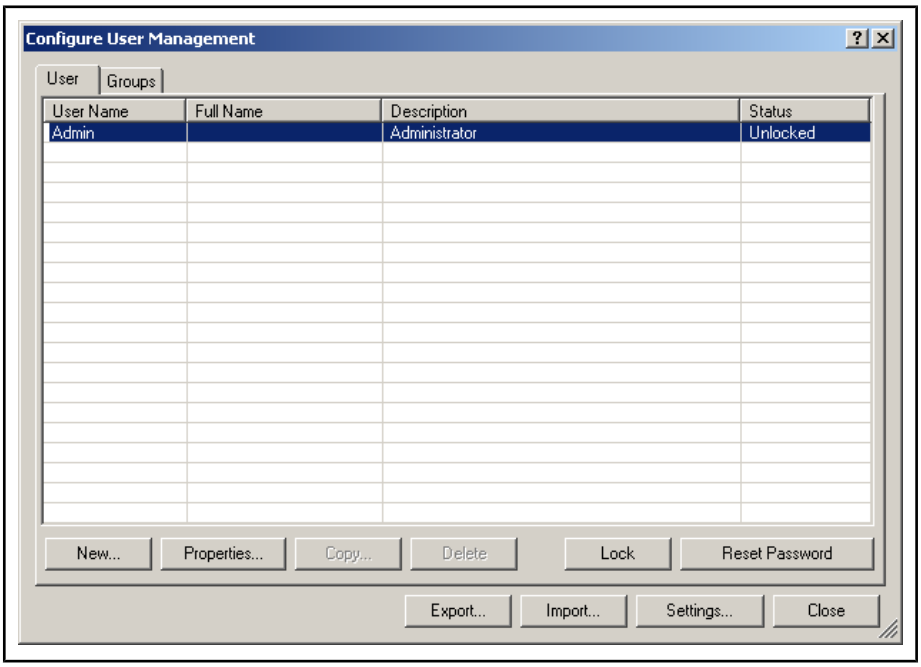


Fig.2-115: Configure User Management dialog box, user list

User name The user name is the name being entered in the login dialog, together with the password.

Full Name This column displays the full name of the user, in the form <last name>, <first name>.

Description This column contains a short description of the user.

State The status of the user is displayed here. Status “unlocked” means, the user can be logged in in IndraWorks. If the status is “locked”, IndraWorks does not allow to login this user.

Below the user list, there are the buttons “New”, “Properties”, “Copy”, “Delete”, “Lock” and “Reset Password”. To select one of those functions click on the according button. The functions are described in detail later in this section.

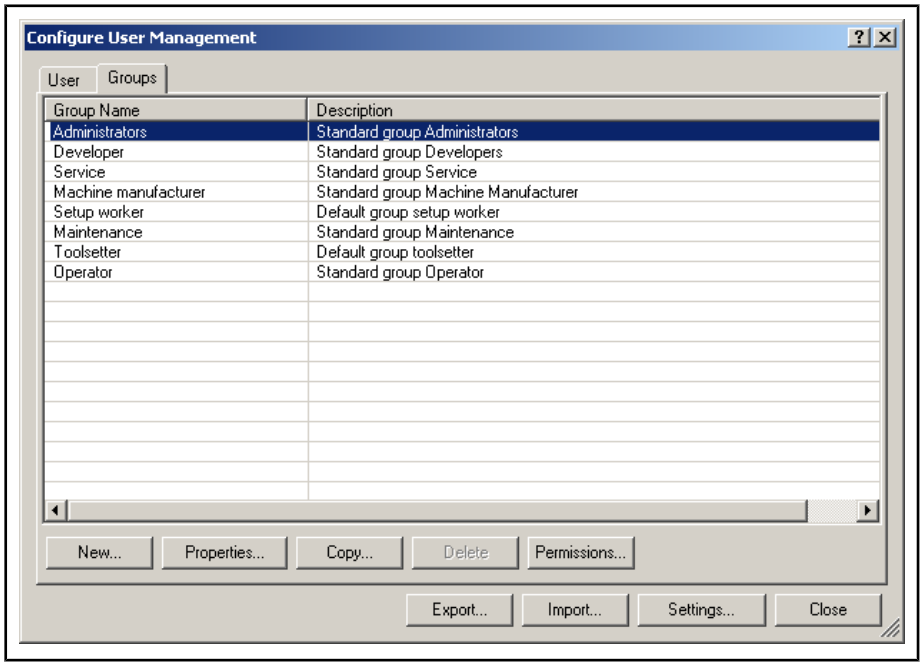


Fig.2-116: Configure User Management dialog box, group list

The group list consists of the columns “Group Name” and “Description”.

Below the group list, there are the buttons “New”, “Properties”, “Copy”, “Delete” and “Permissions”. To select one of those functions click on the according button. The functions are described in detail later in this section.

Creating a User

1. To call the “New User” dialog box open the “Configure User Management” dialog box, go to the “User” tab, and click on the “New” button.

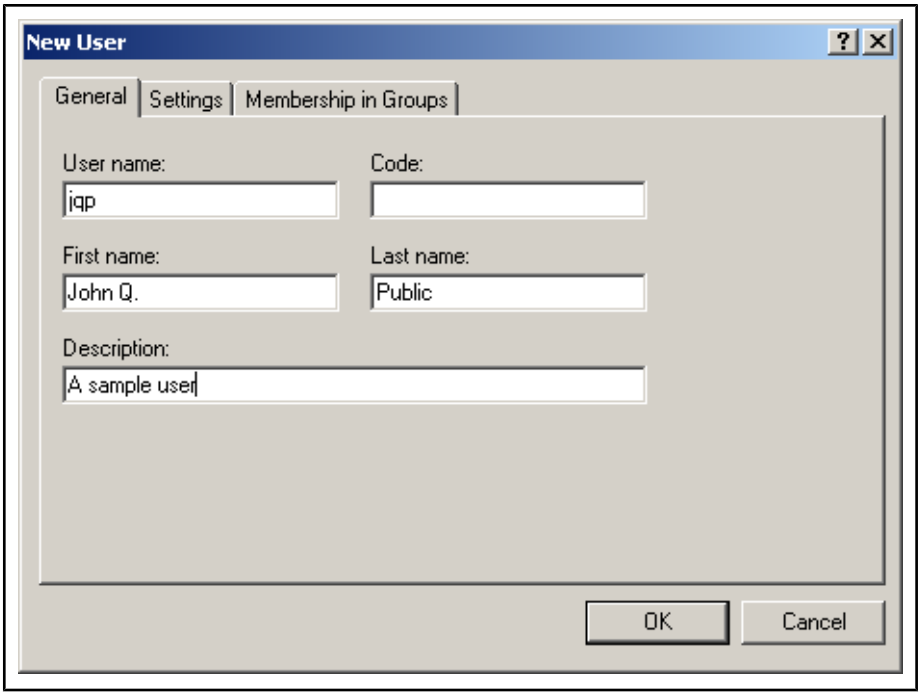


Fig.2-117: New Group dialog box, data entry

User name The user logs in under this name in IndraWorks. It must not contain blanks and it must be unique in the IndraWorks project.

First name First name of the user (optional)

Last name Last name of the user (optional)

Description Brief description of the user (optional)

Code If working with external login procedures, the code assigned to the user must be entered here, e. g. the EKS key. Entered codes must be unique in the project. This field can remain empty.

Enter a user name in the “General” tab. The user name must be unique in the IndraWorks project. You will obtain an error message if the user name has already been assigned to another user. The other input boxes of this tab page are optional and can remain empty.

2. Click on the “Settings” tab and adjust here the settings for the new user.

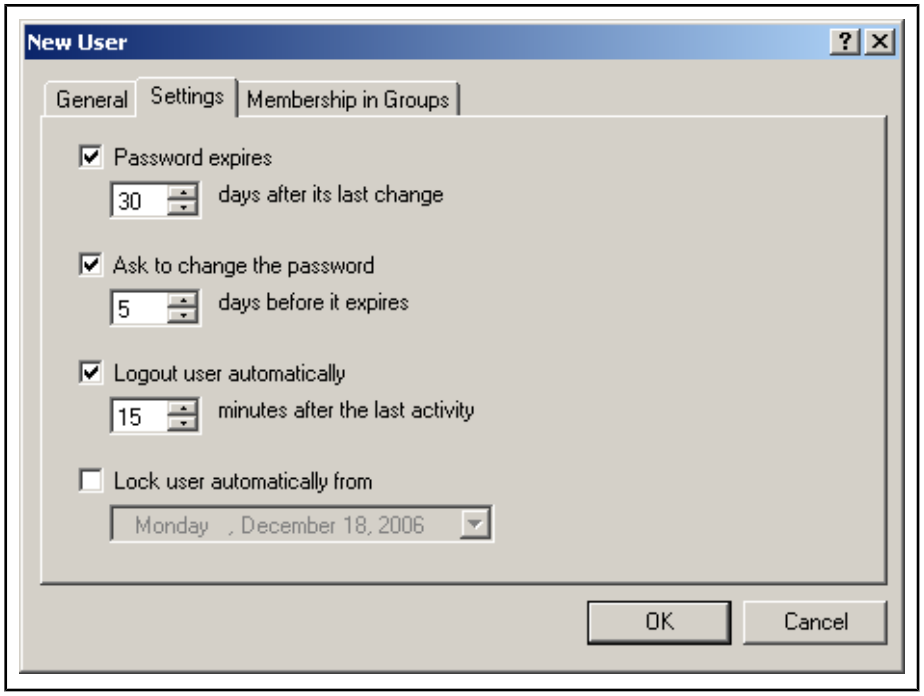


Fig.2-118: New Group dialog box, adjust the settings

Password expires Select this option, if the password of the user is to expire after a defined time. The user will have to enter a new password. You can select a number of days for the expiration time.

Ask to change the password You can select this option if the “Passwor expires” option is selected. If you select “Ask to change the password” the user will be asked to change his password some days before the password expires. You can define the number of days.

Logout user automatically Select this option, to logout the user automatically if he does not perform any actions (keyboard, mouse) for some time. Define a number of minutes for the time from the last action to the automatic logout of the user.

Lock user automatically from Select this option, to lock the user from a defined date on. After this locking the user cannot login in IndraWorks.

3. Click on the “Membership in Groups” tab and assign the new user to one or several groups.

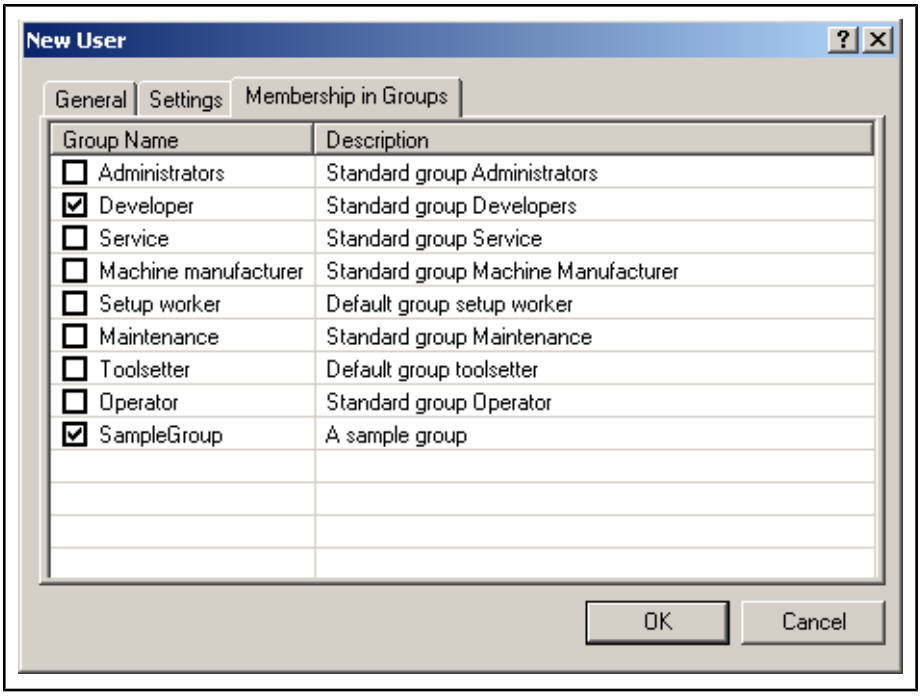


Fig.2-119: New User dialog box, assign new user to groups

4. Click on “OK” to confirm the settings and to close the New User dialog box. The new user appears in the user list now.

Editing a User

1. Open the “Configure User Management” dialog box and select the user to be edited in the user list.
2. Click on the “Properties” button or use <Return> to open the Edit User dialog box. You can also double-click on the user to be edited. The input boxes of the Edit User dialog box contain the current data of the user.
3. Change the data of the user and apply these changes via “OK”.

Copying a User

1. Open the “Configure User Management” dialog box and select the user to be copied in the user list.
2. Click on the “Copy” button to open the Edit User dialog box. The input boxes of the Edit User dialog box contain the data of the original user. But the input boxes for user name and code are empty, because those data have to be unique for every user in an IndraWorks poject.
3. Enter a new user name and, if necessary, change the predefined settings. Confirm these changes via “OK”. The copied user appears in the user list now.

Deleting a User

1. Open the “Configure User Management” dialog box and select the user to be deleted in the user list.
2. Click on the “Delete” button or press <Del>.
3. Click on “Yes” to confirm the following safety prompt. The user will be deleted and the according entry will be removed from the user list.

Locking or Unlocking a User

You can lock a user in order to disable a user account temporarily. The user management declines to login locked users. To enable a user account, you can unlock a locked user anytime. The status locked or unlocked is displayed for each user in the “Status” column of the user list.

To lock or unlock a user proceed as follows:

1. Open the “Configure User Management” dialog box and select the user to be locked or unlocked in the user list. Depending on the state of the selected user the according button below the user list is named “Lock” or “Unlock”.
2. Click on “Lock” or “Unlock” to change the status of the user. You can also change the status using the space bar. The changed status will be displayed immediately in the user list.

Resetting the Password of a User

The Reset Password function changes the password of a user to its initial state. On the next login, the user will be prompted to enter a new password. This function is useful, if a user cannot remember his password and thus cannot login.

1. To reset the password of a user open the “Configure User Management” dialog box and select the user in the user list.
2. Click on the “Reset Password” button. Now, the password of the user is reset. The user will be prompted to enter a new password on the next login.

Creating a Group

1. To call the Edit Group dialog box open the “Configure User Management” dialog box, go to the “Groups” tab, and click on the “New” button.

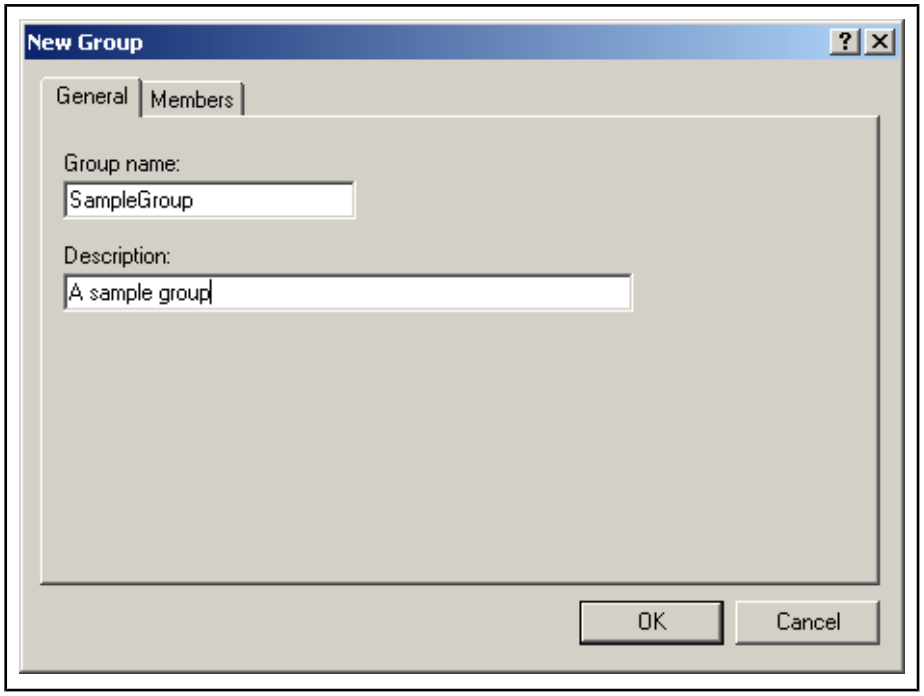


Fig.2-120: New Group dialog box, data entry

Group name Name of the group

Description Brief description of the group (optional)

Enter a group name in the “General” tab. The “Description” input box of this tab is optional and can remain empty.

2. Click on the “Members” tab and assign users to the new group by checking the appropriate check boxes.

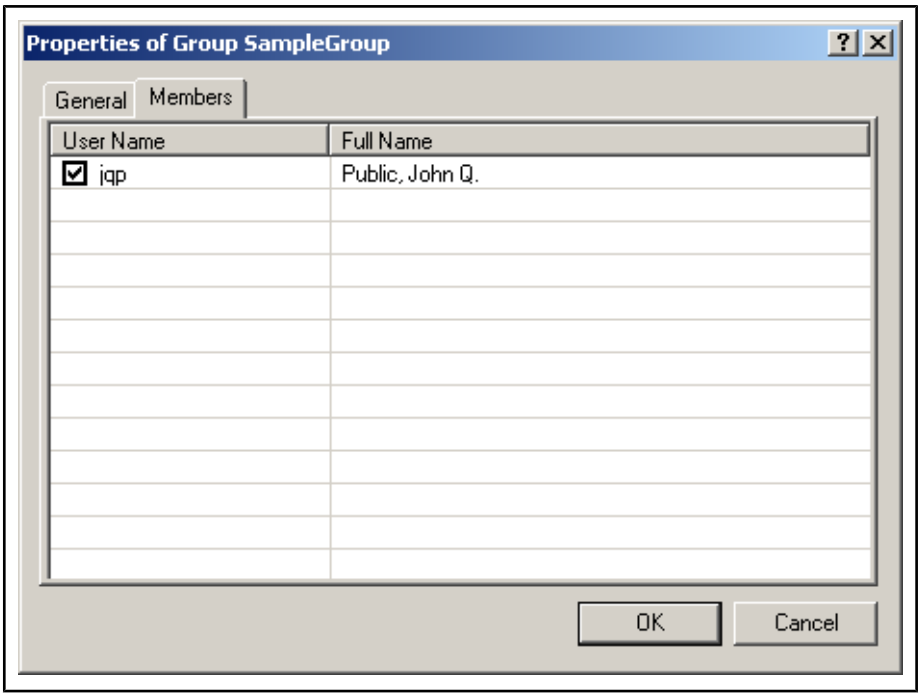


Fig.2-121: New Group dialog box, select members

3. Click on “OK” to confirm the settings and to close the New Group dialog box. The new group appears in the group list now.

Editing a Group

1. Open the “Configure User Management” dialog box and select the group to be edited in the group list.
2. Click on the “Properties” button to open the Edit Group dialog box. The input boxes of this dialog box contain the current data of the group.
3. Change the data of the group and apply these changes via “OK”.

Copying a Group

1. Open the “Configure User Management” dialog box and select the group to be copied in the group list.
2. Click on the “Copy” button to open the Edit Group dialog box. The input boxes of this dialog box contain the data of the original group, except the group name. A new name must be entered.
3. Enter a new group name and, if necessary, change the predefined settings. Confirm these changes via “OK”. The copied group appears in the group list now.

Deleting a Group

1. Open the “Configure User Management” dialog box and select the group to be deleted in the group list.
2. Click on the “Delete” button or press <Del>.
3. Click on “Yes” to confirm the following safety prompt. The group will be deleted and the according entry will be removed from the group list.

Editing the Permissions of a Group

1. Open the “Configure User Management” dialog box and select the group whose permissions you want to edit.
2. Click on the “Permissions” button or use <Enter> to open the Edit Permissions dialog box.
3. Change the permissions of the group by selecting or deselecting the checkboxes on the different tabs.
4. Click on “OK” to confirm the settings and to close the Edit Permissions dialog box.

2.8.5 Settings

Open the “Configure User Management” dialog box and click on the “Settings” button. Now you can define the settings for login, network passwords, network user locking, external login, and logging in the dialog.

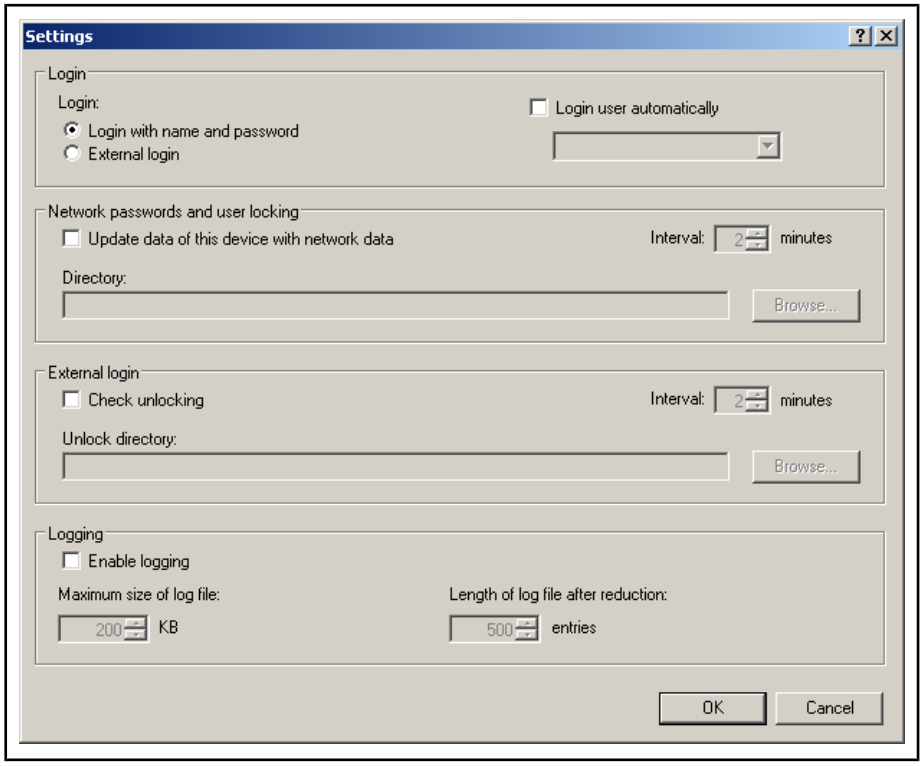


Fig.2-122: Settings dialog box

Login

In the section “Login”, adjust the settings for the login procedure and for the auto login.

Select “Login with name and password” if the users are to enter their user name and their password on login. If users call protected functions without being logged in or without the necessary permissions, the login dialog will be opened automatically.

If an external login procedure is to be used select “External login”. External login applies to all login procedures, that need, e. g., a key switch or an Euchner EKS key, instead of a user name and a password. This enables a login on operation stations without keyboard, simply by inserting a key or an EKS key.

Use the “Login user automatically” option to define a default user. If this option is enabled and a user is selected, the user management causes the login of this default user immediately after the start of IndraWorks or if no other user is logged in.

Network Passwords and User Locking

Select this option, if your IndraWorks project comprises several operation stations and if password changes and user lockings are to be valid for all operation stations of the project.

Select a directory that is enabled for network sharing. This directory is used to synchronize the passwords and the list of locked users in the operation stations.

The selected interval defines the time, after that the synchronzation takes place. The shorter this time, the earlier a change in one operation station effects the other operation stations. However, too short intervals may stress the system unnecessary. Thus a shortest interval of 2 minutes is defined. Select the interval as long as possible and as short as necessary.

External Login

This section only applies to you, if you are using external login procedures and Euchner EKS keys and the Euchner EKM software. The EKM software generates the EKS keys unlock files from the serial number. These files are used by the user management to recognize whether an EKS key is unlocked for a user login.

Select whether the unlocking of the EKS keys is to be checked and which directory contains the unlock files. This directory must be accessible via a network share. Please, enter an interval, after that the unlock files will be updated. Note for this interval: as long as possible, as short as necessary.

Logging

The user management possesses a logging function, that writes all login and logout procedures and all permission enquiries into a log file. Select the “Enable logging” checkbox to switch on the logging function.

The log file is located in the IndraWorks project (...projectdirectory\db \IndraWorks.Userman.Log.csv) and is a unicode CSV text file. It can be opened and analyzed, e. g. by MS-Excel or OpenOffice-Calc. The column separator is a semicolon.

To avoid an uncontrollable growing of the log file, its size will be checked. If the file exceeds a maximum size it will be reduced to a minimum line number. The first entries of the log file will be removed, and the current logging entries will be preserved.

Define via “Maximum size of log file” which size the log file can reach before it is reduced. Use “Length of log file after reduction” to define how many lines will be preserved in the reduced log file.

2.8.6 Export and Import

Exporting a User Data Base

To use the user data base of one IndraWorks project in other IndraWorks projects, export this user data base into a zip file. Then, load the other IndraWorks project(s) and import the zip file with the user data.

1. Open the “Configure User Management” dialog box and click on the “Export” button to export the user data base.
2. Now select the directory, where the user data base is to be exported to and enter the name of the zip file.
3. Start the export via “OK”.

Importing a User Data Base

1. Open the “Configure User Management” dialog box and click on the “Import” button to import an exported user data base into an IndraWorks project.
2. Select the according zip file, containing the data to be imported, in the file selection dialog, and click on “OK” to start the import.

☞ On import the data of the current user data base will be replaced by the data imported. You cannot undo this process. Please, be careful to avoid data loss.

Import ACC001.DAT

Using the import function, you can load user data bases, created with older IndraWorks versions (file ACC001.DAT).

1. Open the “Configure User Management” dialog box and click on the “Import” button to import an ACC001.DAT file into an IndraWorks project.
2. Select the “Acc001.DAT” file type and the file.
3. Start the import via “OK”.

☞ On import the users of the ACC001.DAT file will be loaded. The data of the current user data base will be replaced by the data imported. You cannot undo this process. Please, be careful to avoid data loss.

2.9 Login with EKS-Keys

2.9.1 General

The following sections describe the login using EKS keys.

2.9.2 Activating the EKS System in IndraWorks

Once IndraWorks has been installed, all files and drivers, necessary to operate the EKS system, are available on your PC or control panel, but they are not activated.

To activate the EKS system in IndraWorks proceed as follows:

1. Scan for the “_DDP.OPDLoadingSequenceForEksLoginDC.xml” file in the “Config” sub-directory of the IndraWorks installation directory and delete the underline in front of the file name.
2. If you are using the USB version of the EKS terminals, install the EKS USB driver as described in the following section. If you are using only EKS terminals with serial interface this step is not necessary.

   ☞ You can operate several EKS terminals simultaneously without difficulty; also a mixed operation serial/USB is possible.
3. Restart IndraWorks.
4. Select the external login in the user management.

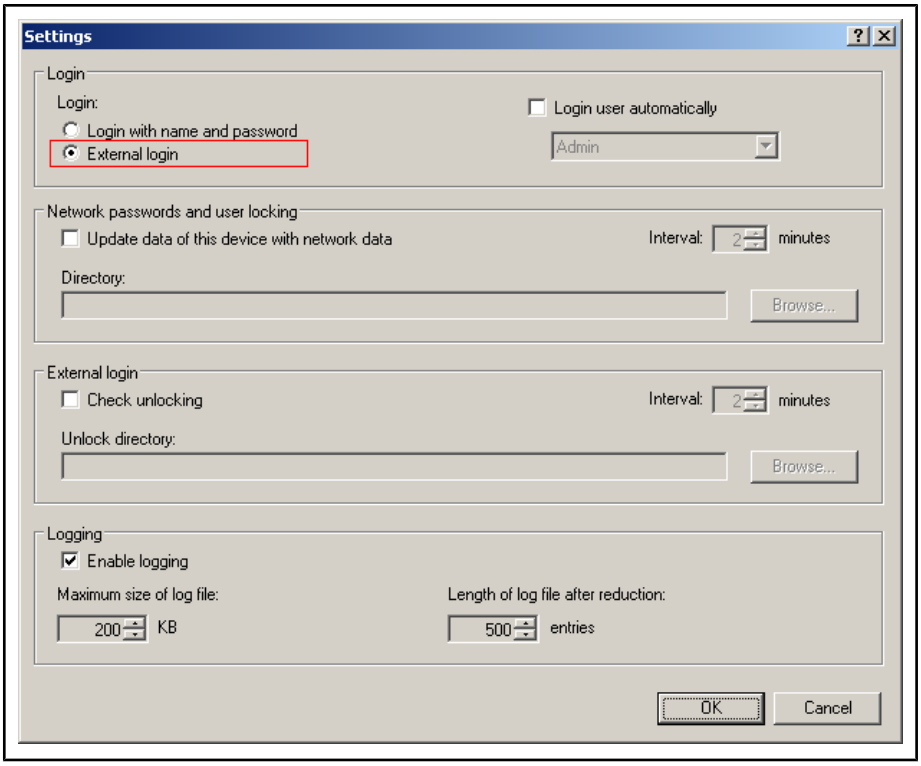


Fig.2-123: Settings, select external login

After selecting the external login you can still login via name and password (**Project ▸ Login User**). Additionally, the user management allows now to login by inserting an EKS keys in a connected EKS terminal. Removing the EKS key from the EKS terminal logs out the user automatically.

2.9.3 Installing the EKS USB Driver

The installation program for the EKS USB driver has been stored on your PC or control panel during the IndraWorks installation. Hence no further data carriers are necessary to install the driver. On control panels you will find the installation program for the driver in the `C:\Programme\IndraWorks\Redist\EuchnerEksUsbDriver` directory.

To install the Euchner EKS USB driver, proceed as follows:

1. Connect the EKS terminal to a free USB port.

   Afterwards, the following message box appears:

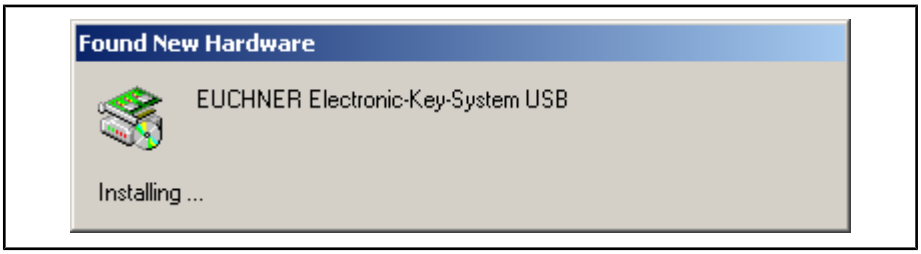


Fig.2-124: Windows recognizes the newly connected EKS terminal

The Windows hardware wizard will be started.

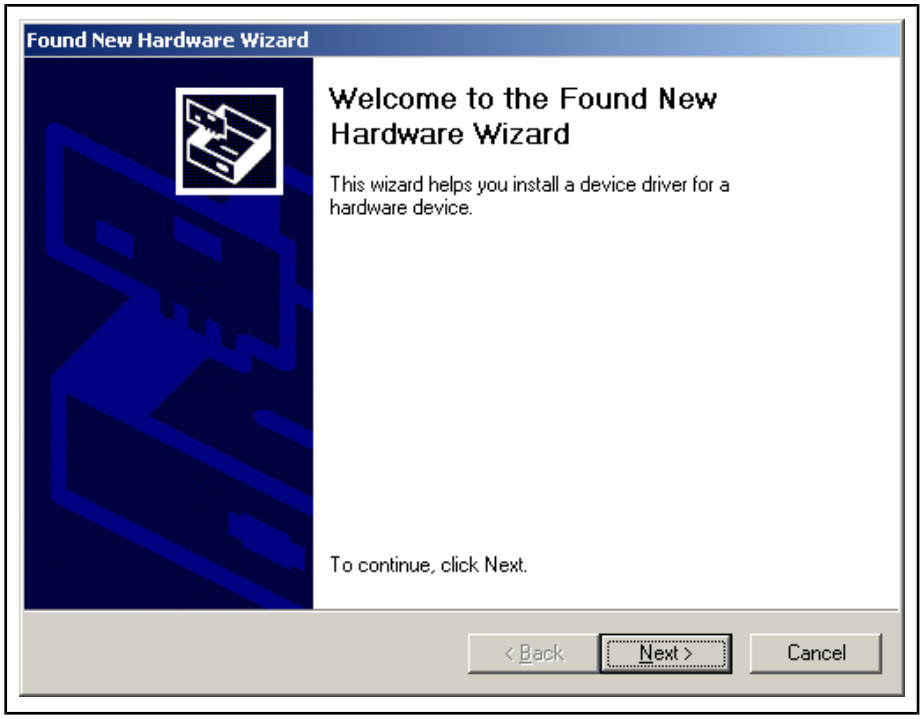


Fig.2-125: Windows hardware wizard

2. Press “Next>>”.

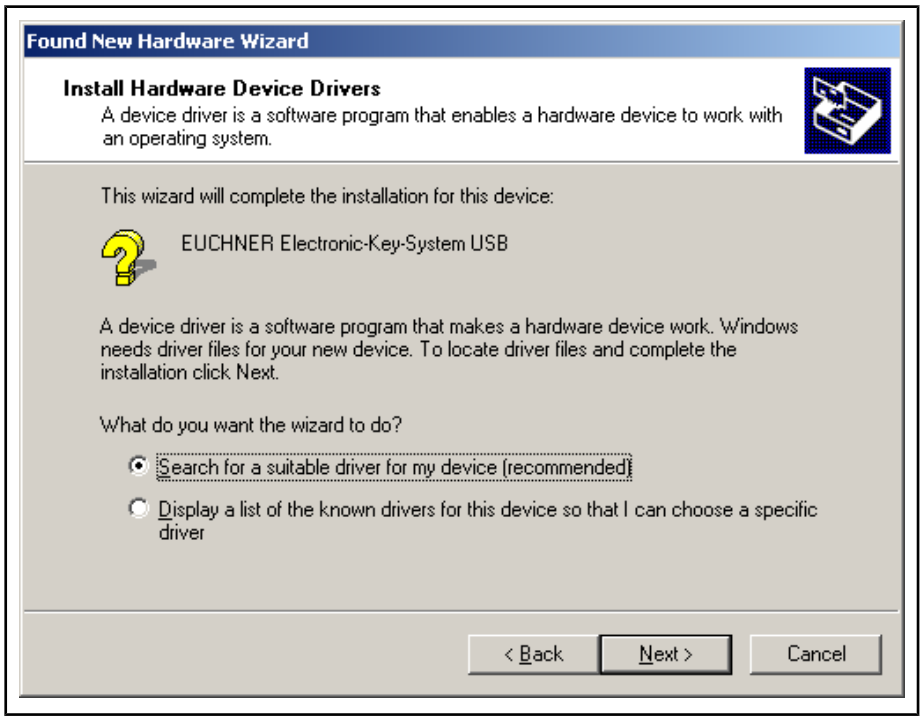


Fig.2-126: Install hardware driver for new virtual serial port

3. Press “Next>>”.

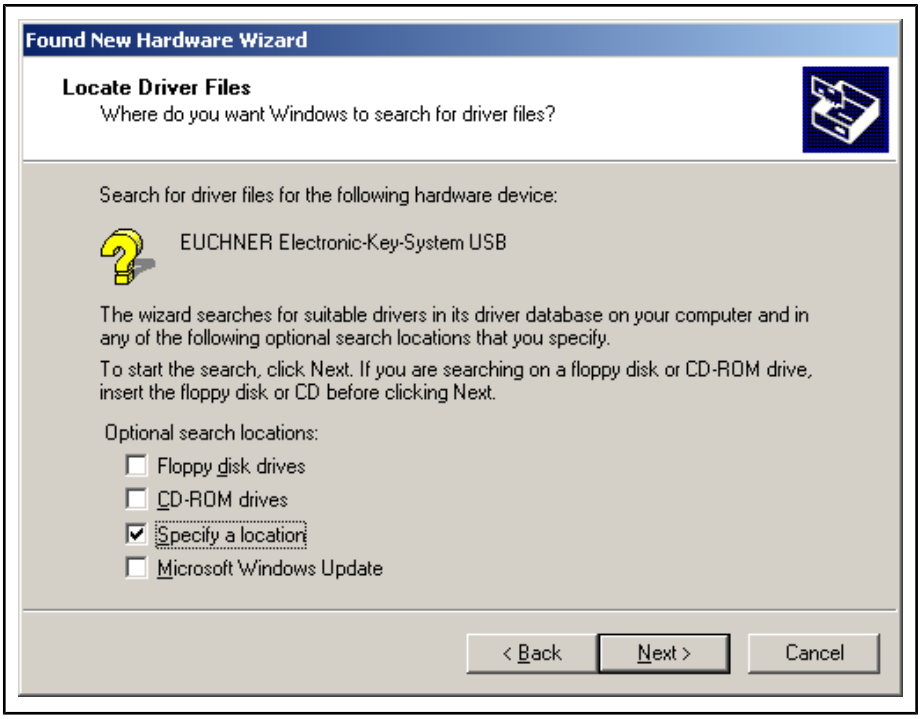


Fig.2-127: Scan for driver files for new virtual serial port

4. Press “Next>>”.

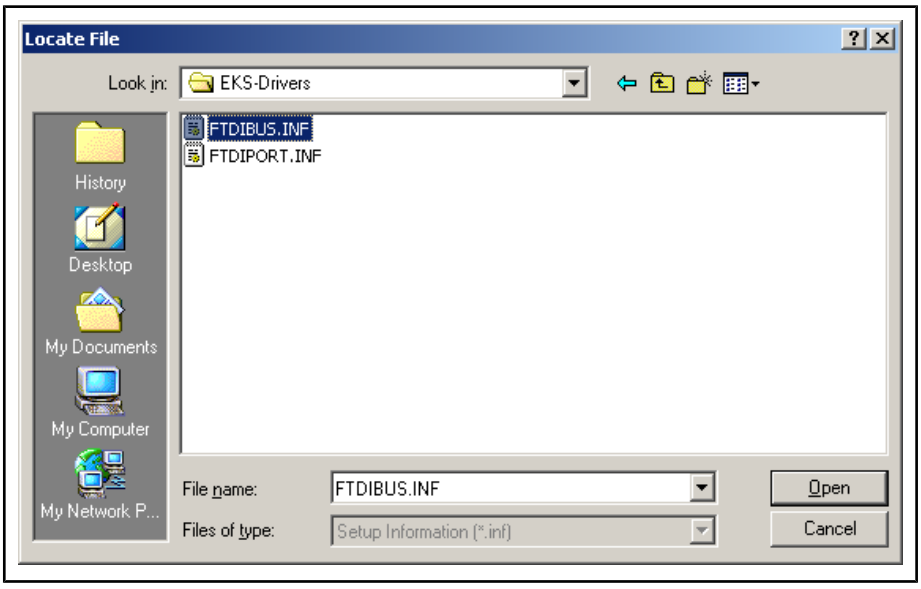


Fig.2-128: Select a driver for EKS terminal

Select the “FTDIBUS.INF” file in the “C:\Programme\IndraWorks\Redist \EuchnerEksUsbDriver” directory and click on “Open”.

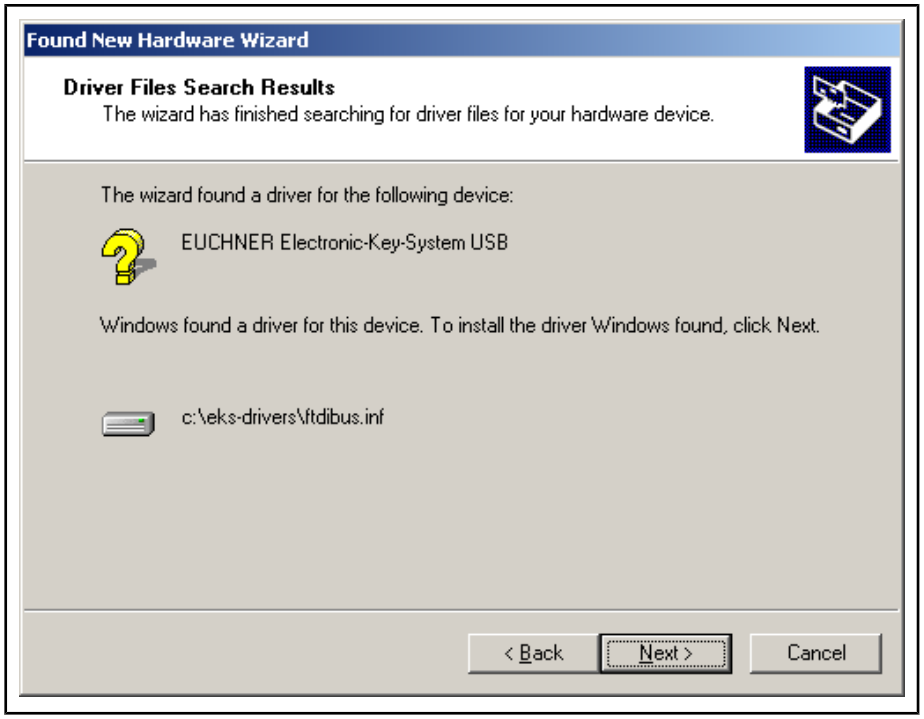


Fig.2-129: Confirm driver selection for new virtual serial port

5. Press “Next>>”.
   The EKS USB driver will be installed now.

Fig.2-130: Windows hardware wizard, complete the installation of the EKS USB driver

6. Click on “Finish” to finish the installation of the EKS USB driver. Immediately after the installation of the EKS USB driver, a virtual serial port will be installed for the newly connected EKS terminal.

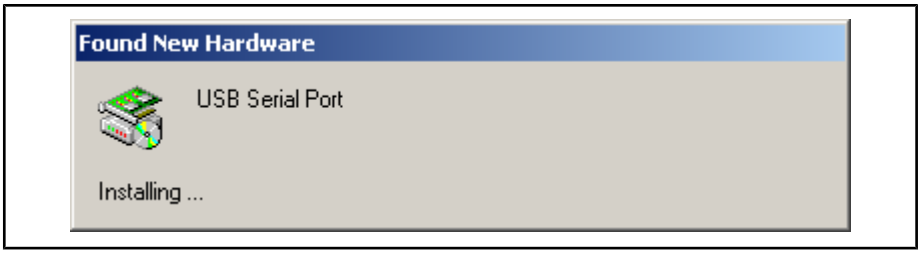


Fig.2-131: Windows recognizes a new virtual serial port

Having recognized the new virtual serial port, Windows starts the hardware wizard again.

Fig.2-132: Windows hardware wizard

7. Press “Next>>”.

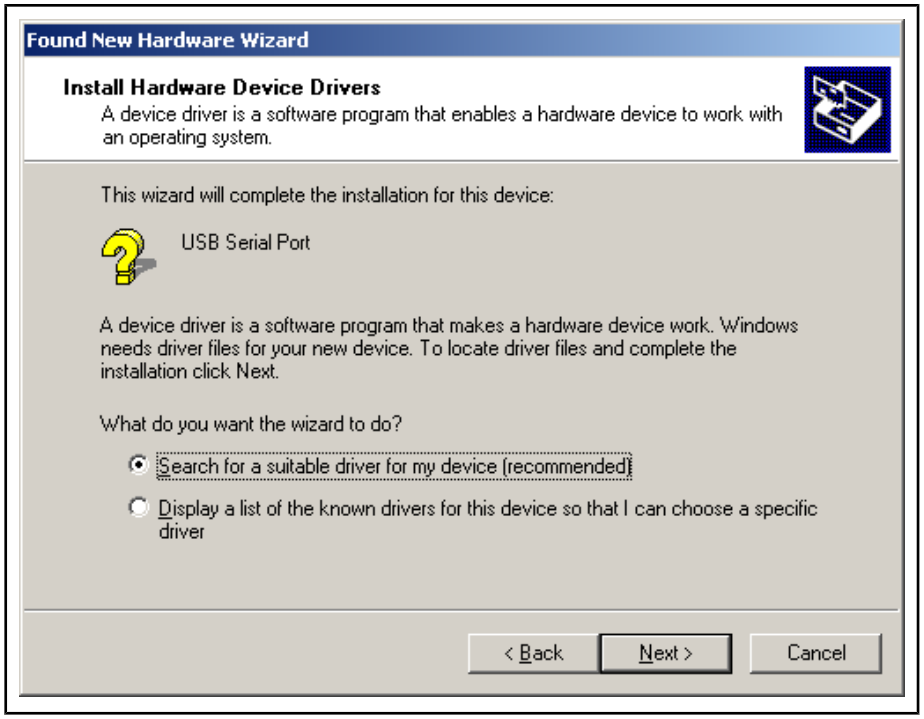


Fig.2-133: Install hardware driver for new virtual serial port

8. Press “Next>>”.

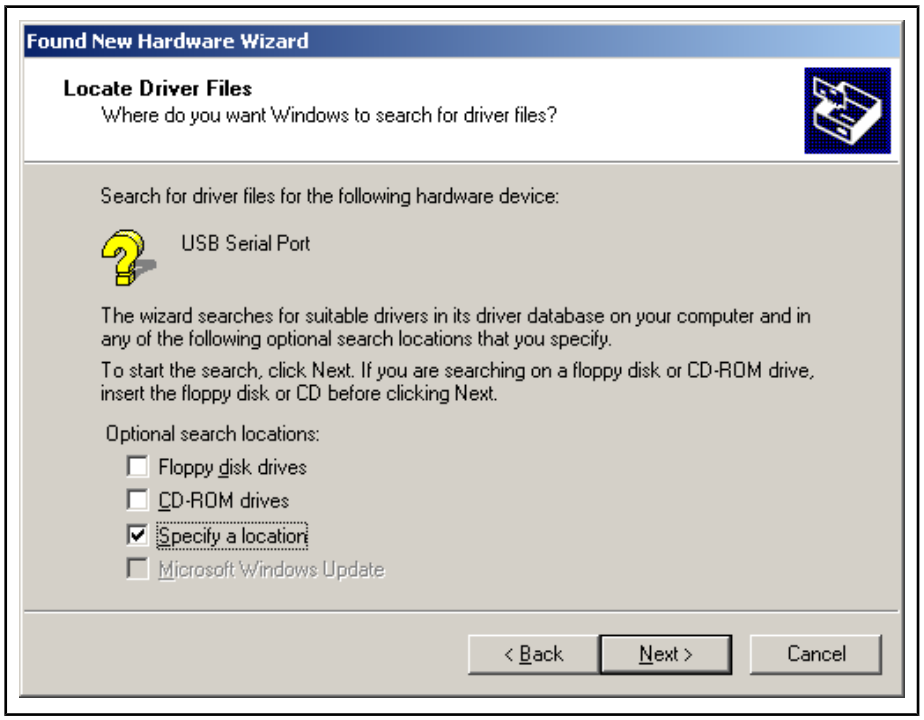


Fig.2-134: Scan for driver files for new virtual serial port

9. Press “Next>>”.

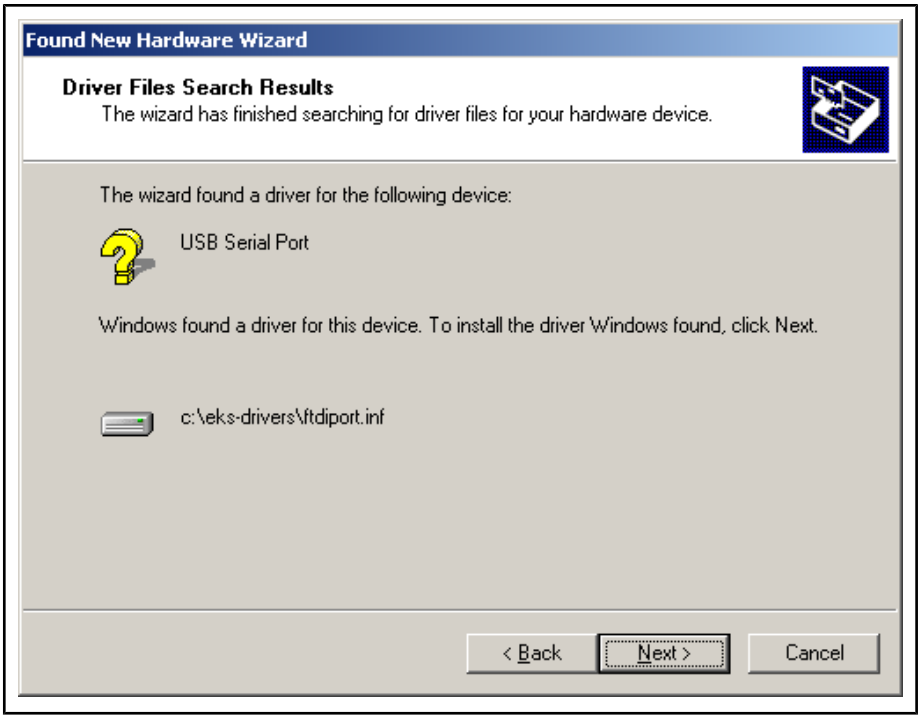


Fig.2-135: Confirm driver selection for new virtual serial port

10. Press “Next>>”.

Fig.2-136: Finish the installation of the virtual serial port

11. Press “Finish”. Now the installation of the driver is completed and the EKS terminal is ready.

2.9.4 Creating Users and Groups

The IndraWorks user management analyzes the protection level stored in the EKS key as well as the special function “Rework” and logs in a user, depending on these data. The user to be logged in will be defined via a code given by the administrator.

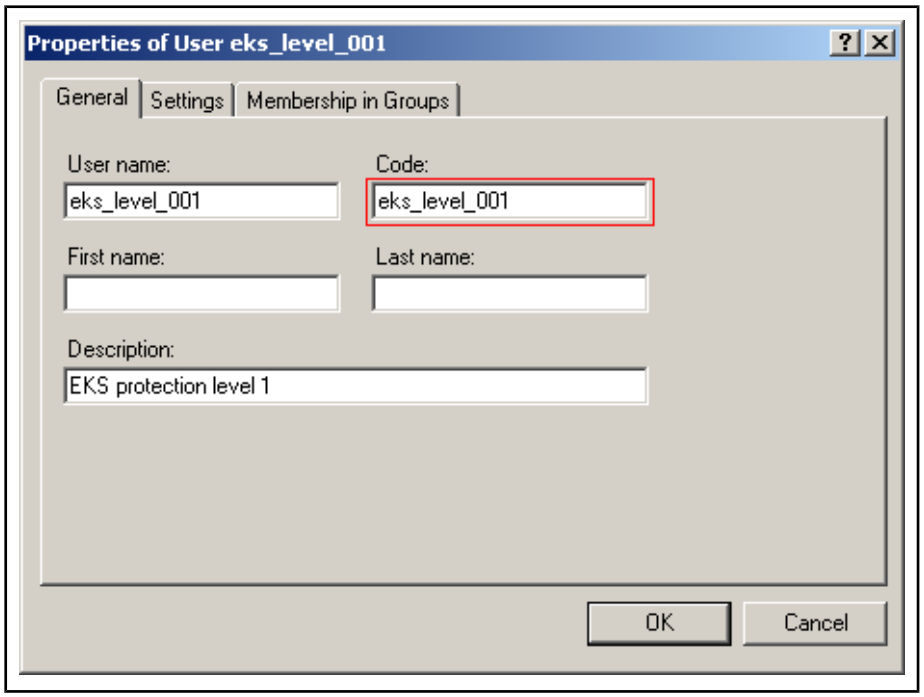


Fig.2-137: Properties dialog box, Code defines the user to be logged in with a certain EKS key

The user management forms the code from the EKS key data, according to the following scheme:

Pattern: eks_level_<protectionlevel>[_[r][o]]

The protection level is a 3-digit decimal number. The user management supports protection levels from 000 to 099. _r or _o will be added if the special function Rework or OperatingSystem is set on the EKS key.

Here you will find some examples of codes, formed from EKS key data:

- eks_level_001 : EKS key with protection level 1, no special function
- eks_level_002 : EKS key with protection level 2, no special function
- eks_level_001_r : EKS key with protection level 1, special function Rework
- eks_level_001_o : EKS key with protection level 1, special function OperatingSystem
- eks_level_001_ro : EKS key with protection level 1, special functions Rework and OperatingSystem

We recommend the following user configuration and group configuration when using the EKS system in IndraWorks:

1. Create users for any protection level and users for necessary protection levels with special functions. Assign a code to the users, according to the protection level and special functions.

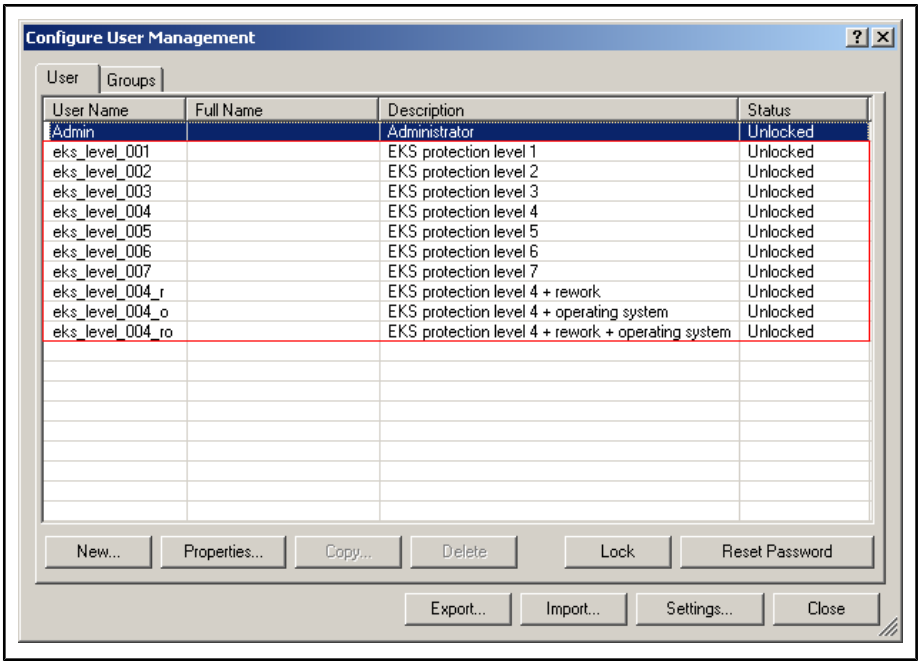


Fig.2-138: User scheme according to EKS protection levels

2. Create a group for any protection level and for both special functions. Assign the necessary permissions to these groups. If the permission profiles of the groups are based on each other, assign only the additional permissions to the higher groups.

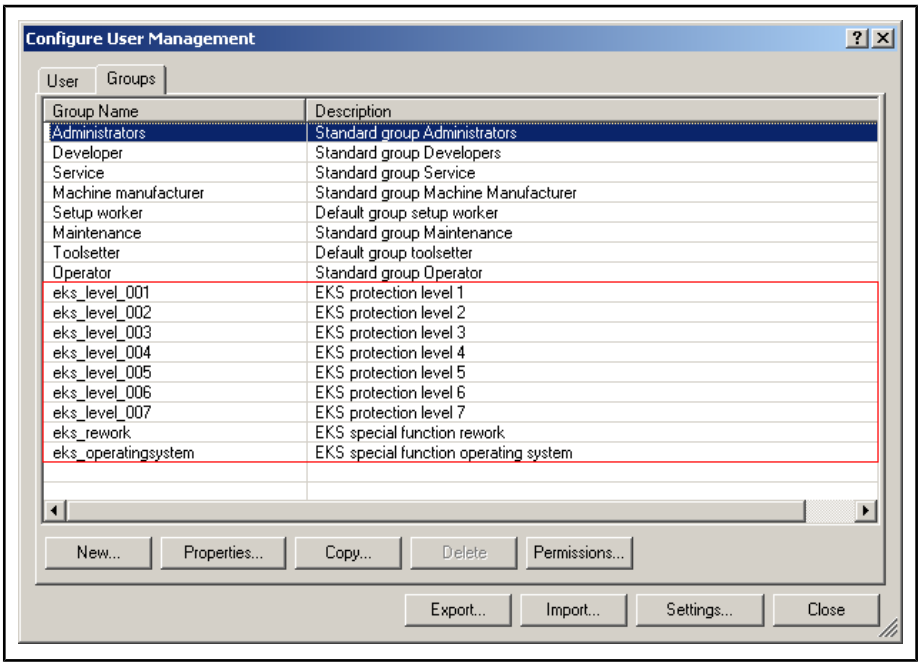


Fig.2-139: Group scheme according to EKS protection levels and special funktions

3. Assign all the new users to the appropriate groups.

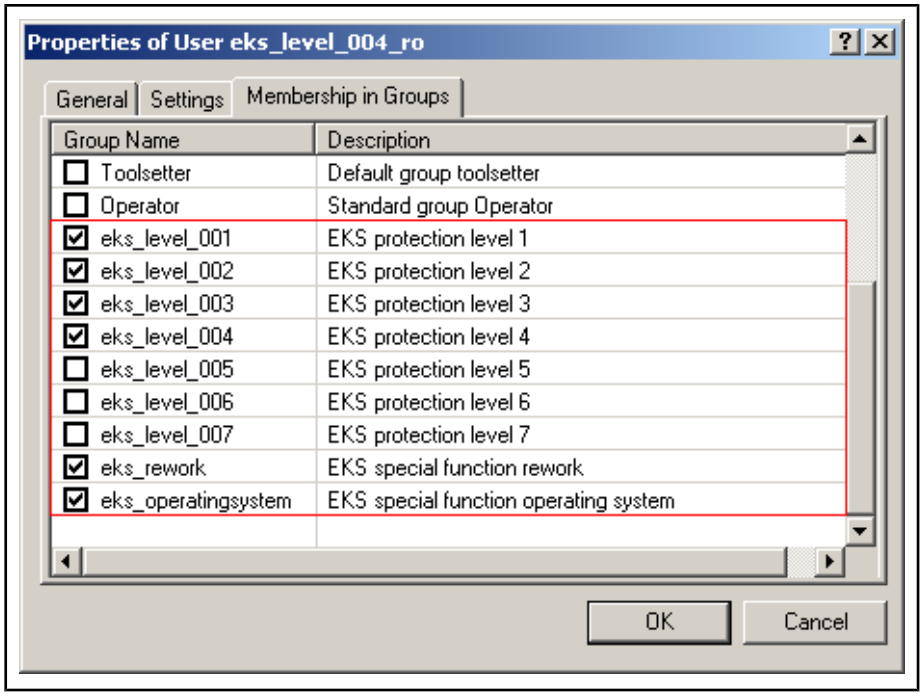


Fig.2-140: Properties dialog box, users of an EKS protection level are members in the according groups. Here EKS protection level 4 with special funktions Rework and OperatingSystem.

2.9.5 Reading the EKS Key Data in the PLC Program

In order to evaluate EKS key data or information on the current protection level in an PLC program, define the following global PLC variables:

Program:

```
VAR_GLOBAL
    EKS_SerialNo:STRING(16);      (*key serial number*)
    EKS_ProtectionLevel:INT;      (*protection level*)
    EKS_WorkerIdent:STRING(7);    (*worker ident*)
    EKS_SafetyOperationMode:BYTE; (*safety mode*)
    EKS_So1:BYTE;   (*special functions*)
    EKS_So2:BYTE;   (*special functions*)
    EKS_So3:BYTE;   (*special functions*)
    EKS_So4:BYTE;   (*special functions*)
    EKS_So5:BYTE;   (*special functions*)
    EKS_So6:BYTE;   (*special functions*)
    EKS_So7:BYTE;   (*special functions*)
    EKS_So8:BYTE;   (*special functions*)
    EKS_So9:BYTE;   (*special functions*)
    EKS_KeyIn:BYTE; (*FFhex if key inserted, 00hex if no key*)
END_VAR
```

As soon as an EKS key has been inserted into the EKS terminal and the according user has been logged in, IndraWorks will write the following data in all PLC devices belonging to the current project, if the variables above have been declared.

EKS_SerialNo Serial number of the EKS key, as string. If the EKS key will be removed, this variable is an empty string.

EKS_ProtectionLevel Protection level, stored on the EKS key. Protection levels from 00 to 99 are allowed. If the EKS key will be removed, this variable will be -1.

EKS_WorkerIdent	Workerident (e. g. personnel number), as string. You can use this data field to personalize EKS keys. If the EKS key will be removed, this variable is an empty string.
EKS_SafetyOperationMode	Safety mode. If the EKS key will be removed, this variable will be 0.
EKS_So1 to EKS_So9	Special functions. If the EKS key will be removed, this variables will be 0.
EKS_KeyIn	Key inserted flag. As long as an EKS key is inserted and an according user is logged in, this variable is FFhex, otherwise it will be 00hex.

2.10 PLC Interface of the User Management

2.10.1 General Information

The following sections describe the interface between user management and PLC. This interface allows to control the user-login and user-logout by the PLC, and to read the logged in user in the PLC program.

2.10.2 Activating the PLC Interface

Once IndraWorks has been installed, all files, necessary to operate the PLC interface, are available on your PC or control panel, but they are not activated.

To activate the PLC interface in IndraWorks proceed as follows:

1. Exit IndraWorks
2. Scan for the “_DDP.OPDLoadingSequenceForPlcLogin.xml” file in the “Config” sub-directory of the IndraWorks installation directory and delete the underline in front of the file name.
3. Select the external login in the user management of IndraWorks, if you want to login and logout users via the PLC.

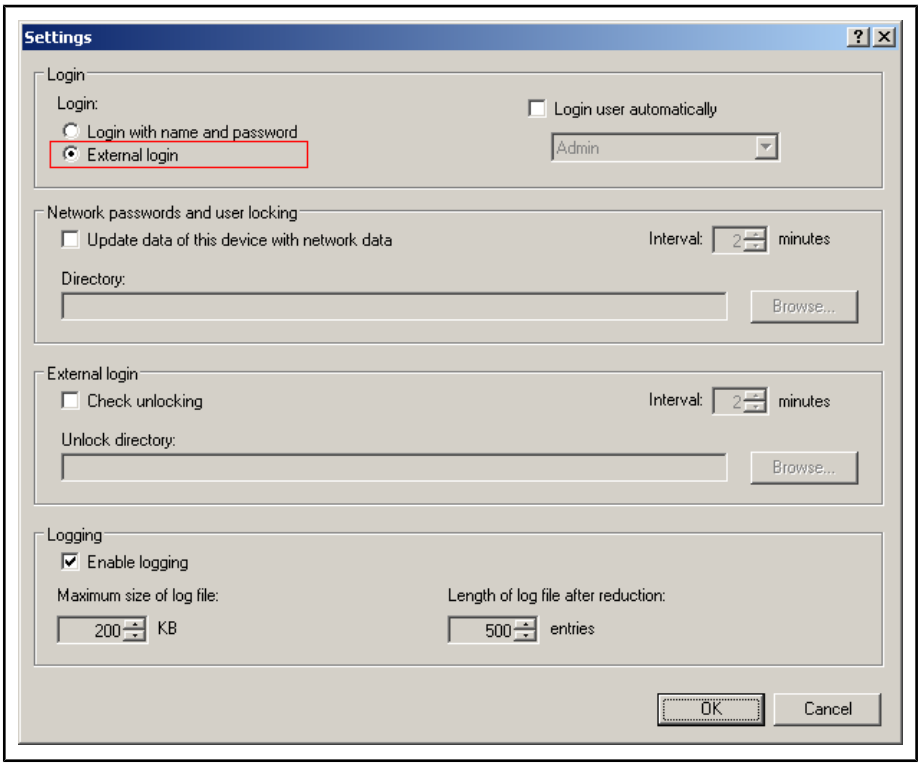


Fig.2-141: Settings, select external login

> ☞ After selecting the external login you can still login via name and password (**Project ▸ Login User**). Additionally, the user management allows to login and logout users by programming certain PLC variables.

4. Restart IndraWorks.

2.10.3 Login and Logout Users via a PLC Program

The user login and user logout is controlled by three PLC variables. Define the following global PLC variables:

Program:

```
VAR_GLOBAL
    UM_LoginCmd:BYTE;    (* command, 00hex = logout, FFhex = login *)
    UM_LoginCode:STRING; (* code of the user to be logged in *)
    UM_MediaId:DWORD;    (* release code, can be set to 0  *)
END_VAR
```

To login a user, the PLC program has to perform the following steps:

1. Set PLC variable UM_MediaId to 0
2. Write the code of the user to be logged in in the PLC variable UM_LoginCode
3. Set PLC variable UM_LoginCmd to FFhex

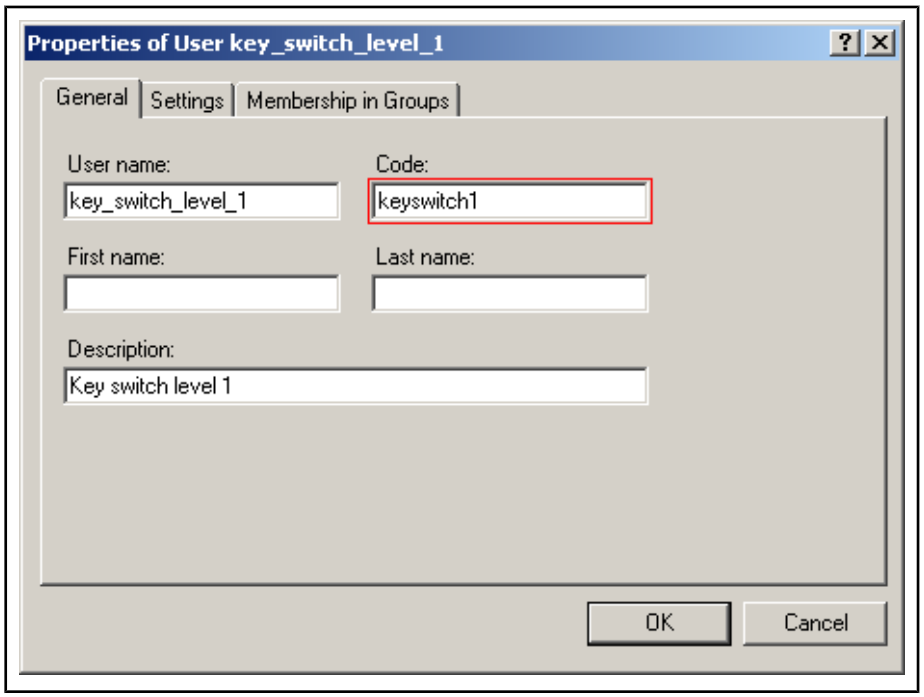


Fig.2-142: Properties dialog box, defined code must be written in PLC variable UM_LoginCode, to login the user

In order to logout a user, the PLC program must set the PLC variable UM_LoginCmd to 0. This causes the logout of the current user, independent of the values of the other two PLC variables.

2.10.4 Reading the Current User in the PLC Program

The state of the user management is represented by three PLC variables. Define the following global PLC variables:

Program:

```
VAR_GLOBAL
    UM_UsermanEnabled:BYTE;    (*indicates, whether the user management is enabled*)
    UM_UserLoggedIn:BYTE;      (*indicates, whether a user is logged in*)
    UM_CurrentUserName:STRING; (*login name of the user *)
END_VAR
```

The values of these PLC variables are written by the user management. Only reading access to the PLC variables is allowed within the PLC program.

The PLC variable UM_UsermanEnabled indicates, whether the user management is enabled (FFhex) or disabled (00hex). If the user management is disabled, no permissions are asked for in the user interface and all functions are available.

The PLC variable UM_UserLoggedIn indicates, whether a user is logged in (FFhex) or not (00hex).

The PLC variable UM_CurrentUserName contains the user name of the user logged in or an empty string, if no user is logged in.

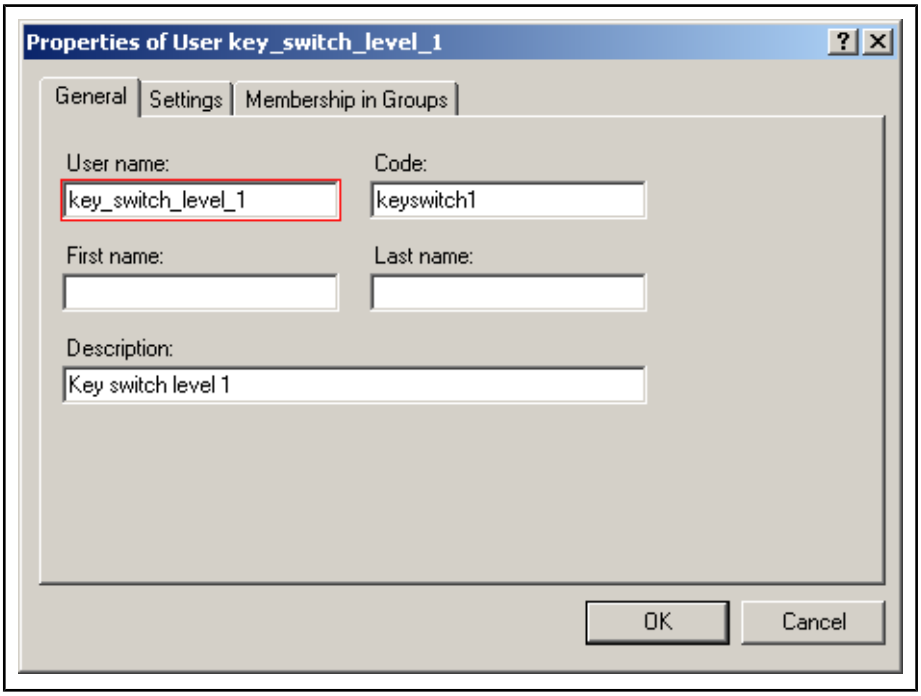


Fig.2-143: Properties dialog box, user management writes the user name in PLC variable UM_CurrentUserName after login

2.10.5 Example: Changing Permission Levels via Key Switch

Task

A key switch connected to the control panel is to be used to switch between several permission levels. The key switch can adopt four positions:

Position 1: Permission level "Operator" In this position the key can be removed, i. e., this permission level is active, even without the key.

Position 2: Permission level "Maintainer"

Position 3: Permission level "Programmer"

Position 4: Permission level "Machine vendor"

Configuration of the User Management

In the user management, one user will be defined for each of these four permission levels. The users will be assigned to one or more groups, according to the necessary permission profile.

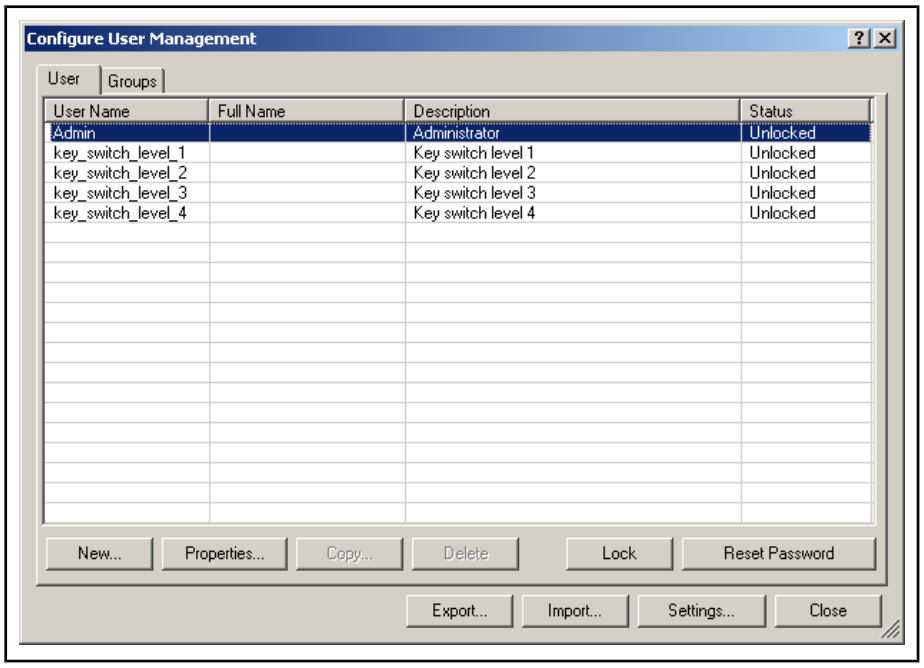


Fig.2-144: Example of a configuration: one user per permission level

The external login will be selected in the settings of the user management. Additionally, the user "key_switch_level_1" is defined here to be logged in automatically. This ensures the availability of at least this permission level .

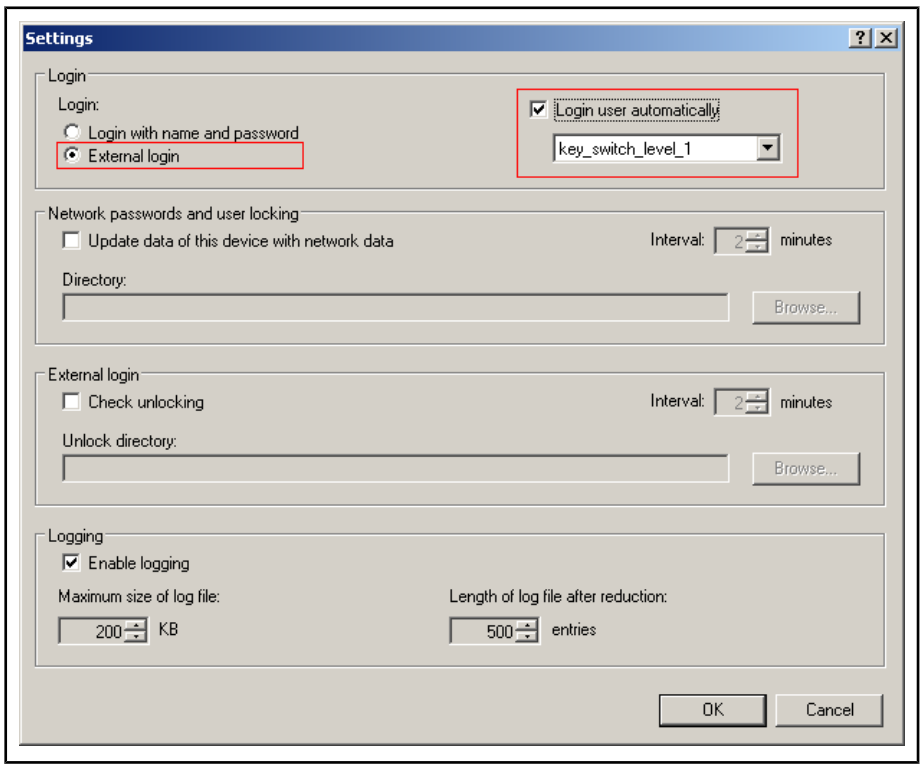


Fig.2-145: Settings, external login

PLC program

The key switch position will be assigned to a user via a PLC program. This program checks the key switch position and logs in the appropriate user.

Program:

```
VAR_GLOBAL
    UM_UsermanEnabled:BYTE;
    UM_UserLoggedIn:BYTE;
    UM_CurrentUserName:STRING;

    UM_LoginCmd:BYTE;
    UM_LoginCode:STRING;
    UM_MediaId:DWORD;
END_VAR
```

Program:

```
PROGRAM PLC_PRG
VAR
    KeySwitch:BYTE; (*key switch*)
END_VAR
```

Program:

```
(*
**  Example:
**  PLC controlled switching of permission levels by key switch
*)

(*
    actual  key inquiry
    The inputs used by the key switch will be copied to the variable keyswitch

    Key switch position 1: 16#01
    Key switch position 2: 16#02
    Key switch position 3: 16#04
    Key switch position 4: 16#08
*)

UM_MediaId := 0; (*release control not used*)

(*Key switch position 4, Bit 4 set*)
IF KeySwitch = 16#08
THEN
UM_LoginCode := 'keyswitch4';
UM_LoginCmd := 16#FF;

(*Key switch position 3, Bit 3 set*)
ELSIF KeySwitch = 16#04
THEN
UM_LoginCode := 'keyswitch3';
UM_LoginCmd := 16#FF;

(*Key switch position 2, Bit 2 set*)
ELSIF KeySwitch = 16#02
THEN
UM_LoginCode := 'keyswitch2';
UM_LoginCmd := 16#FF;

(*Key switch position 1 or key removed, Bit 1 set*)
ELSIF KeySwitch = 16#01
THEN
UM_LoginCode := 'keyswitch1';
UM_LoginCmd := 16#FF;

(*other values: logout current user*)
```

```
ELSE
UM_LoginCmd := 16#00;
UM_LoginCode := '';

END_IF

RETURN;
```

2.11 Firmware Management

The firmware management in IndraWorks provides the following functions:

- Downloading a firmware to a device
- Displaying the current firmware in a device
- Displaying the firmware available for a download

Starting the Firmware Management

To start the firmware management, use the context-sensitive menu of a device or the device-specific entries in the main menu. Select the **Firmware Management** item.

Description of the Dialog Elements

The left selection window of the dialog box displays the firmware available for a download. Depending on the used device it may be possible to select a directory containing firmware files, above this selection window.

The right window of the dialog box displays the firmware of the device.

Below these windows, the current state is displayed, e. g. “Ready for download”.

Operation

Select the firmware you want to load into the device in the left selection window. Depending on the used device further options, e. g., “Reboot after download” may be offered below this window. Start the download via the “Download” button. A progress bar will be displayed, informing on the current state and progress of the download.

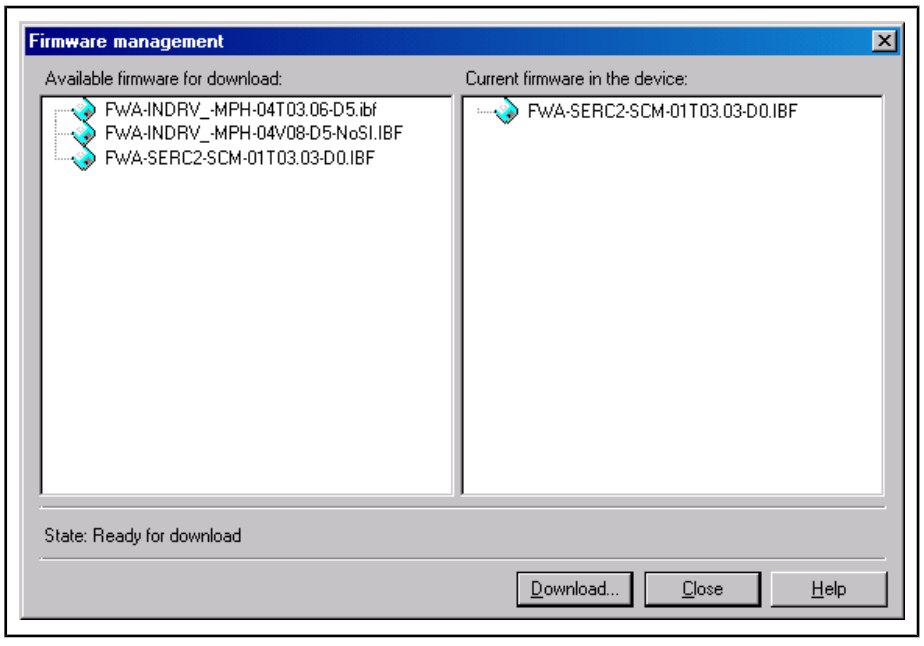


Fig.2-146: Firmware Management dialog box

2.12 License Management

2.12.1 Licensing of IndraWorks Components

General Information

The licenses of IndraWorks software components are managed via the Options dialog box. Open the Options dialog box via **Tools-Options** and then select the “Software Licenses” page in “General”.

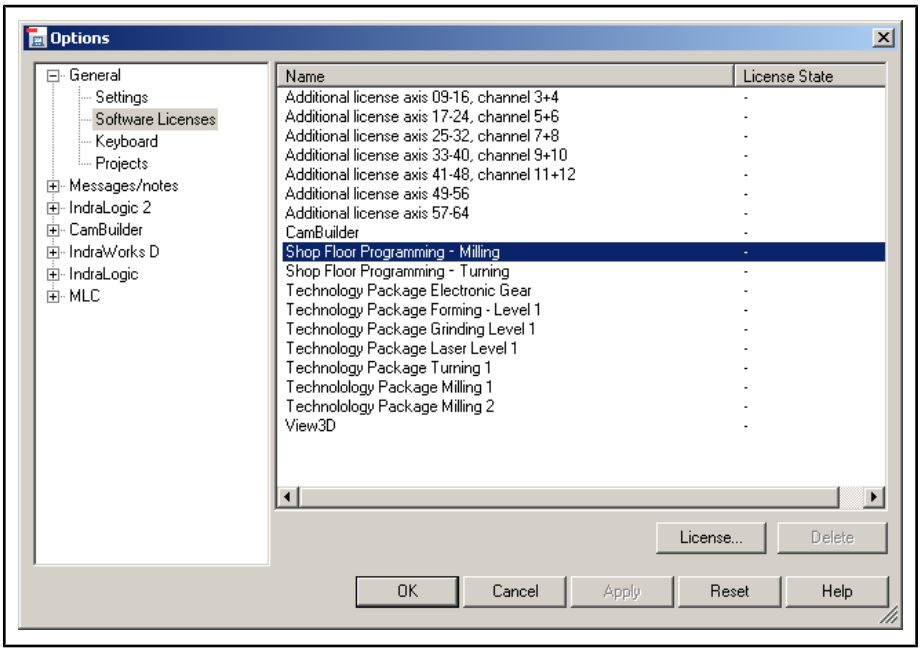


Fig.2-147: Options dialog box, software licenses

The page displays all available software components requiring a license and their license state. The scope of the entries depends on the system installed.

License state	Description
-	No license has been installed on this computer. The component cannot be used.
Licensed	A full license has been installed on this computer. The component can be used without any restrictions.
Demo license, 30-days	A limited demo license has been installed on this computer. The component can still be used for evaluation for the number of days specified (no more than 30 days).

Fig.2-148: License state

To view a description of a license, place the mouse pointer on the license entry.

Installing a License

To license a software component, select it from the list and click on “License...”.

If a full license is already installed, the “License...” button is deactivated.

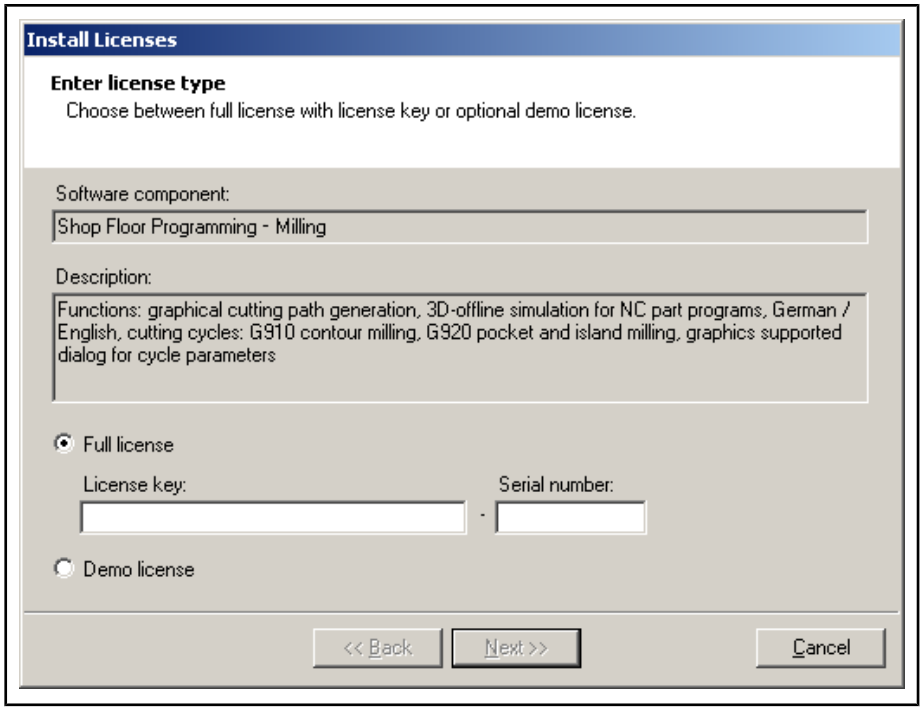


Fig.2-149: Install Licenses dialog box, enter the license type

Decide whether you wish to install an unrestricted full license or a demo license. The demo license allows you to test the selected software components for 30 days.

☞ If “Demo license” is deactivated, a demo license has either already been installed or the component selected does not support a demo license.

For installing a full license, enter the license key and, if displayed, the serial number. You have obtained the necessary data togehter with the license.

Click on “Next>>”.

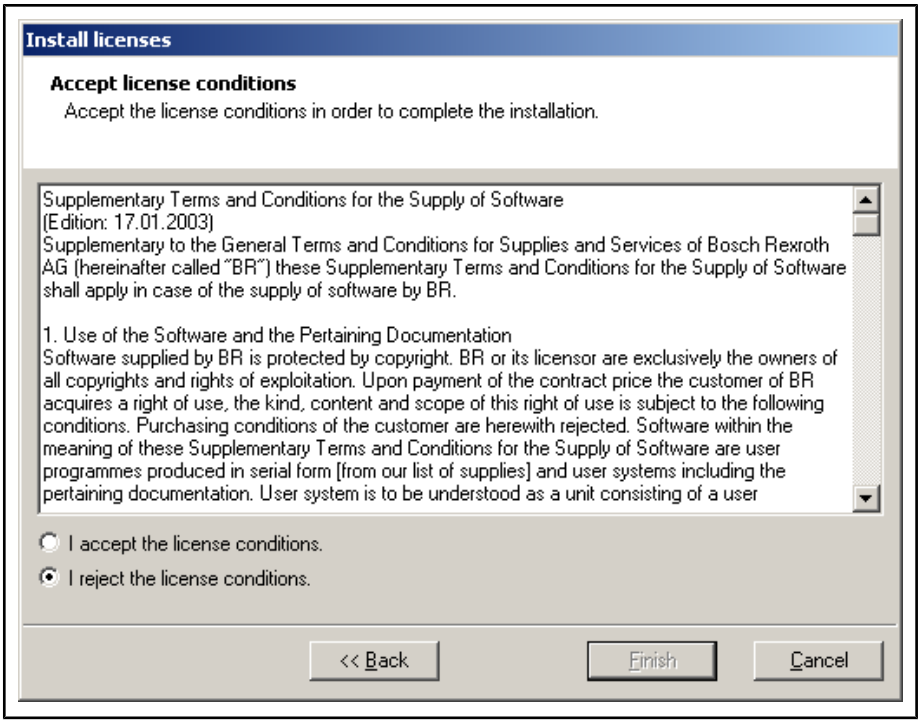


Fig.2-150: Install Licenses dialog box, license conditions

Thoroughly read the license conditions from beginning to end.

To accept the license conditions, select “I accept the license conditions” and click on “Finish”.

If you do not accept the license conditions, click on “Cancel”. In this case, the license will not be installed.

Deleting a License

To delete a license, select it from the list and click on “Delete”.

Fig.2-151: Deleting licenses

To confirm the safety prompt, click on “Yes”.

2.12.2 Licensing Firmware Functions

General Information

Firmware functions requiring a license are enabled via the context-sensitive menu of the particular device in the project explorer. If a device supports functions that can be enabled, it provides the menu item **Firmware Licenses…**.

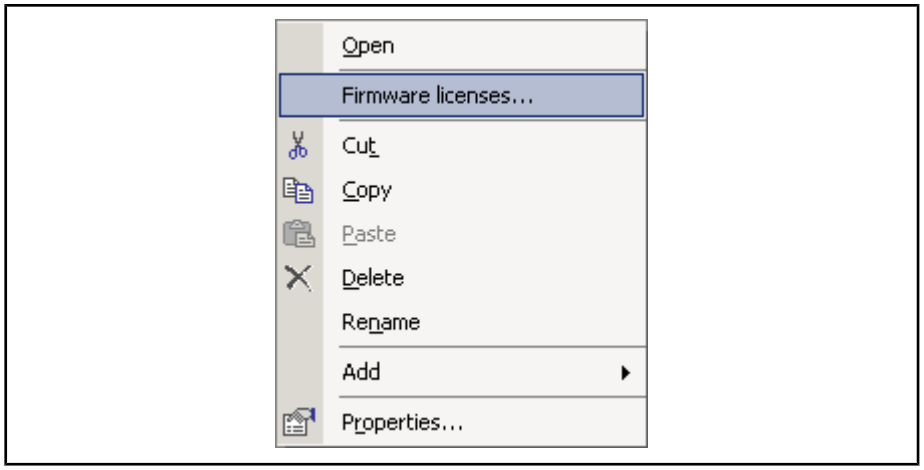


Fig.2-152: Firmware Licenses, menu entry

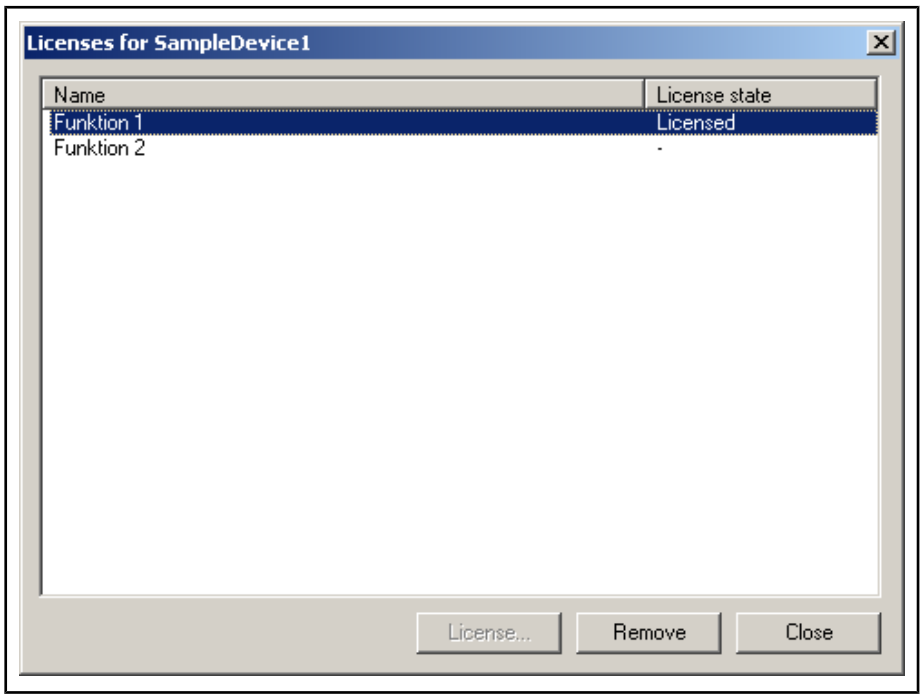


Fig.2-153: Licenses for ... dialog box

The “Firmware Licenses” dialog box displays all functions that can be enabled, together with their license state. The scope of the entries depends on the device type.

License state	Description
-	Function disabled
Licensed	Function enabled

Fig.2-154: License state

To view a description of a function, place the mouse pointer on the corresponding entry.

Enabling Firmware Functions

A license key is not required for enabling firmware functions. However, the functions provided in the “Firmware licenses” dialog box must be paid. You can enable and use those functions, if you have purchased the according licenses.

To enable the functions, select them and click on “License...”.

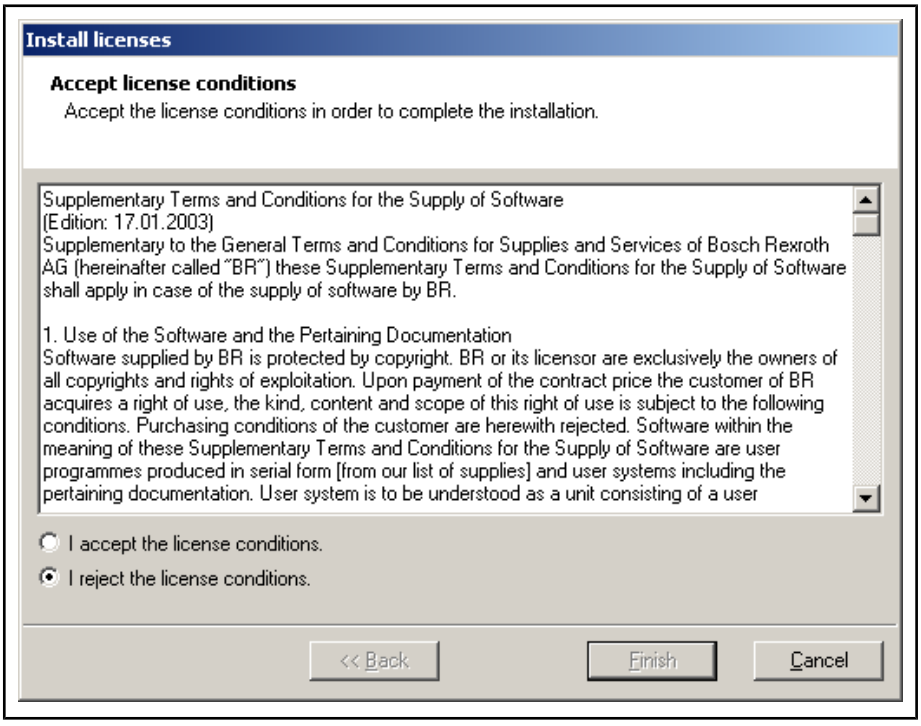


Fig.2-155: Install Licenses dialog box, accept license conditions

Thoroughly read the license conditions from beginning to end.

To accept the license conditions, select “I accept the license conditions” and click on “Finish”.

If you do not accept the license conditions, click on “Cancel”. The functions selected are not enabled.

Deleting Firmware Licenses

To cancel the enabling of functions, select the functions desired and click on “Delete”.

Fig.2-156: Deleting licenses

To confirm the safety prompt, click on “Yes”.

2.13 Network Connection

2.13.1 General Information

To operate several machines in a network with access to data and diagnostics of neighboring machines, the following steps are required:

- A network configuration must be created, that lists all visualization devices and controls of the machines of the network.
- The network configuration must be added to the machine projects.

The data access to a network control is always established via the visualization device to which this control is assigned to. Thus the network configuration must contain the visualization devices as well as the controls.

2.13.2 Creating a Network Configuration

General Information

Create a new network configuration via **File** ▸ **New** ▸ **Network Configuration**. IndraWorks opens an editor, which you can use to edit and save this empty network configuration.

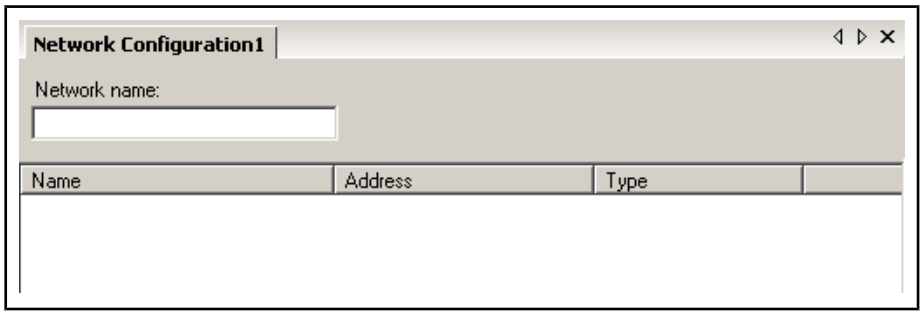


Fig.2-157: Network configuration editor

You can specify a name for the network, and define the network devices in the network configuration editor.

Adding Devices to the Network Configuration

The following devices can be added to a network configuration:

Family	Device Types
Visualization devices	BTV/Vxx
IndraLogic	IndraLogic L20, IndraLogic L40, IndraLogic V, IndraLogic VPP21
MTX	IndraMotion MTX L40, IndraMotion MTX P40, IndraMotion MTX P60

Fig.2-158: Available device types

To add a device to the network configuration, right-click on the device list in the editor and select in **Add** the desired device type.

You can also drag the desired device from the library to the device list (“drag-and-drop”).

If you add a device to the network configuration, a dialog box for editing the device properties will be opened. The device will not be added to the device list unless you have closed this properties dialog box via **OK**.

Tip: It is recommended to define a visualization device for a control, before you add this control (IndraLogic or MTX).

Editing Device Properties

General Information

The properties dialog box is opened automatically when a device is added. To change the properties of a device already contained in the list open the dialog by

- double-click on the device, or
- selecting the device and pressing Enter, or
- selecting the device and **View** ▸ **Properties** in the main menu, or
- right-clicking on the device and selecting **Properties** in the context-sensitive menu.

Visualization Devices (BTV/Vxx)

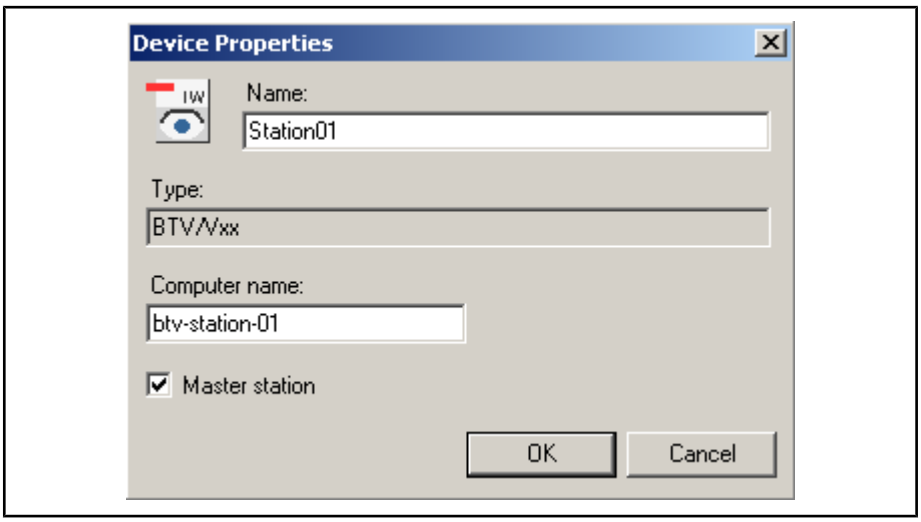


Fig.2-159: Properties dialog box for BTV/Vxx devices

Name Input box for the device name, it must be unique in the network connection. The editor ensures the uniqueness.

Type Device type (display only)

Computer Name Input box for the name used to access the device in the network

Master Station Defines, whether the visualization device is the master station. The editor ensures that only one visualization device becomes the master station.

Controls (IndraLogic, MTX)

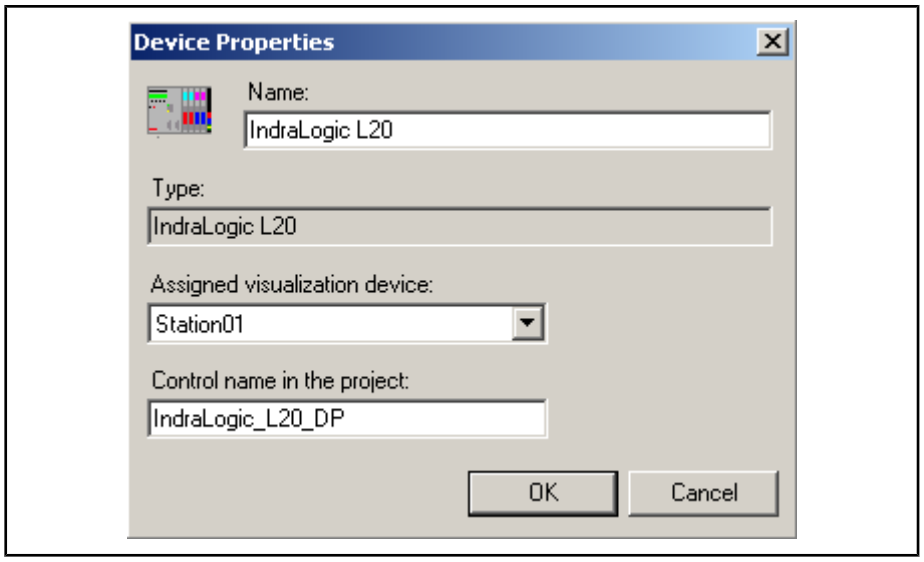


Fig.2-160: Properties dialog for IndraLogic devices and MTX devices

Name Input box for the device name, it must be unique in the network connection. The editor ensures the uniqueness.

Type Device type (display only)

Assigned Visualization Device Defines the visualization device, the control is assigned to. The check box lists all visualization devices defined in the network configuration.

Control Name in the Project Enter the name of the control here, exactly as it has been defined in the original project.

Changing the Order

You can change the order of the devices in the network configuration by

- right-clicking on a device and selecting **Up** or **Down** in the context-sensitive menu, or
- dragging a device to a different position

Removing a Device

You can remove a device from the network configuration by

- right-clicking on the device and selecting **Remove** in the context-sensitive menu, or
- selecting the device and pressing <Entf>, or
- selecting the device and **Edit** ▸ **Remove** in the main menu.

If you are sure you want to remove the device confirm the following note.

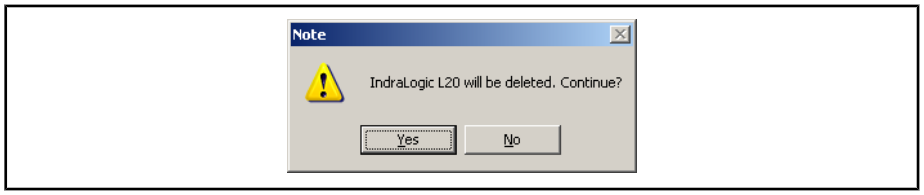


Fig.2-161: Note when removing devices

Saving a Network Configuration

Save a network configuration via **File** ▸ **Save**.

If it is a new network configuration, a dialog for defining the file name will be opened.

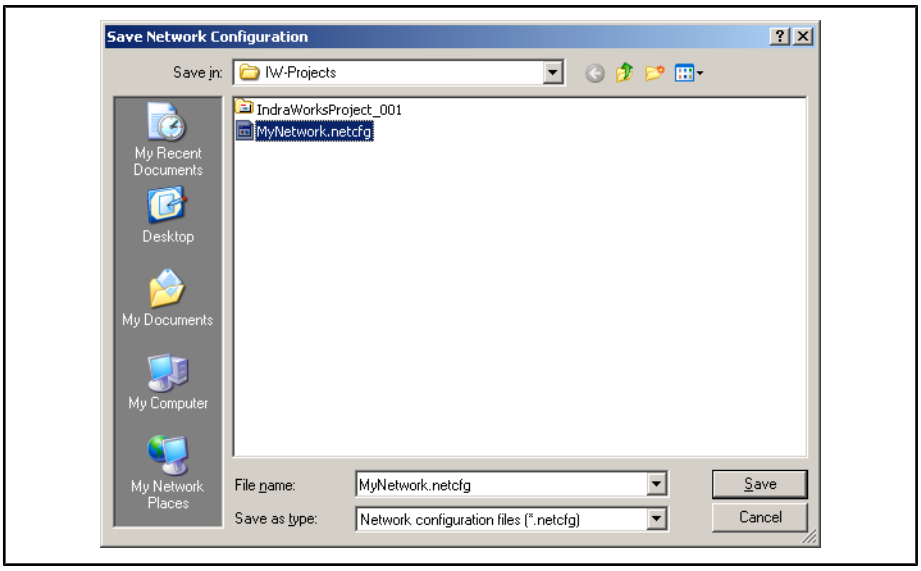


Fig.2-162: Save Network Configuration dialog box

If the network configuration has been saved already or an existing network configuration has been opened, the existing file will be overwritten.

If you want to save the network configuration under a different name, select **File ▸ Save As** in the main menu.

☞ When saving, make sure that the window of the network configuration editor is active, otherwise other data may be saved.

Closing the Editor

To close the network configuration editor, click on the x in the upper right of the window or press <Ctrl> + <F4>.

If there are unsaved changes in the network configuration, you will be asked how to proceed.

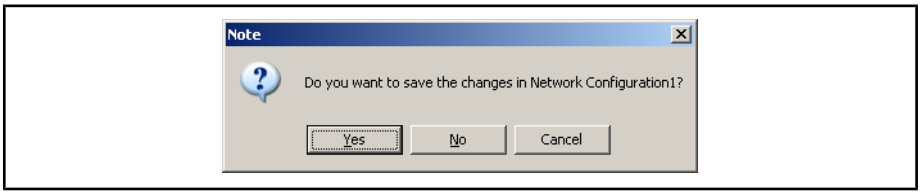


Fig.2-163: Prompt when closing the editor

Yes Saves the network configuration and closes the editor. If the network configuration has not been saved before, the dialog for selecting a file name will be opened.

No Discards the changes and closes the editor.

Cancel The editor will not be closed.

2.13.3 Opening an Existing Network Configuration

Open an existing network configuration via **File ▸ Open ▸ Network Configuration**. IndraWorks opens a dialog box to select the file.

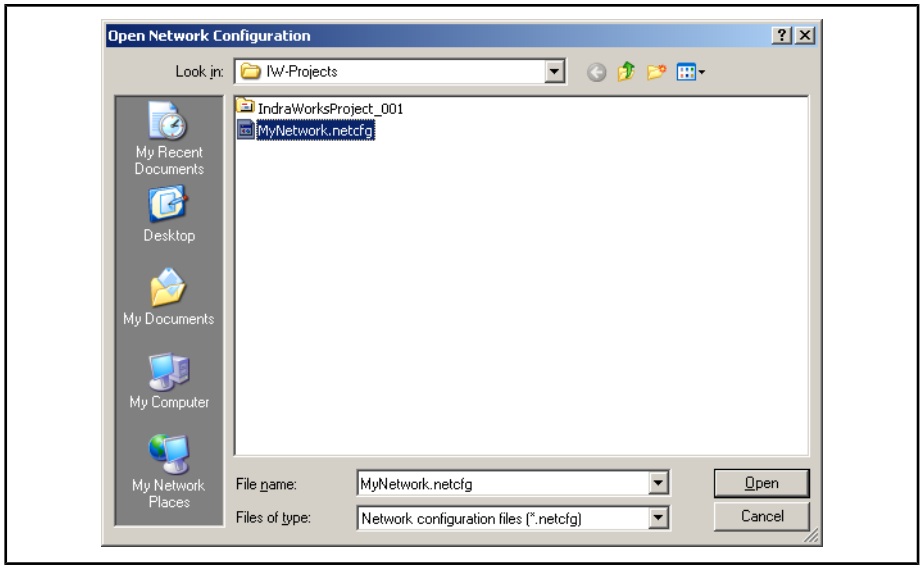


Fig.2-164: Open Network Configuration dialog box

Confirm with Open. The network configuration editor will be opened and displays the projected data.

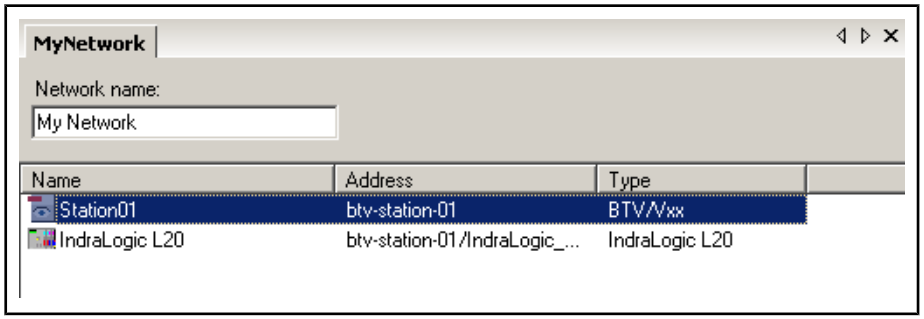


Fig.2-165: Network configuration editor

Edit a network configuration in the same way as described in “Creating a Network Configuration”.

2.13.4 Inserting a Network Configuration in a Project

To display data of non-local controls in a visualization device in operating mode, an according network configuration has to be inserted in the project.

Select **Project ▸ Insert Network Configuration** in the main menu of the open project. IndraWorks opens a dialog for selecting a configuration file.

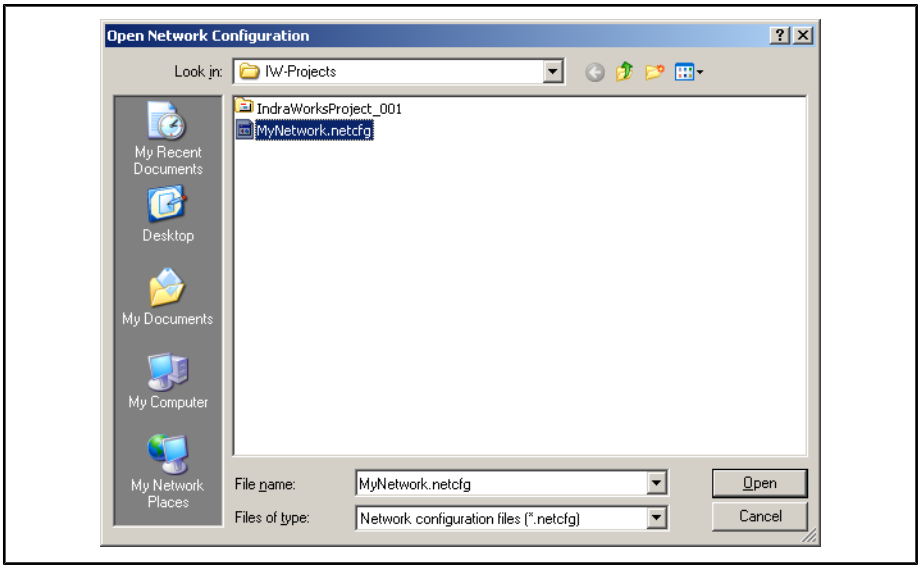


Fig.2-166: Open Network Configuration dialog box

Select a file and press **Open**. The network configuration will be inserted in the project and displayed in the project explorer.

Fig.2-167: Project with network configuration

If you are trying to insert a network configuration into a project that already contains one, you will get the following note.

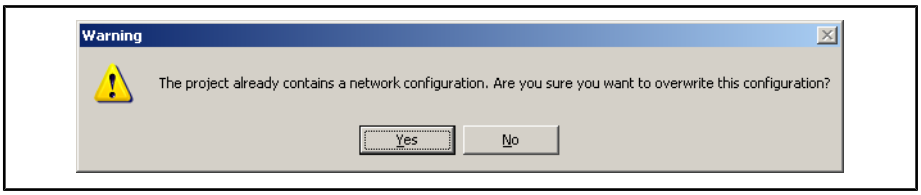


Fig.2-168: Warning when inserting a network configuration into a project

2.14 Remote Engineering

2.14.1 General Information

A project contains all devices, communication connections and other components required for operating a machine or system. A project is operated via the IndraWorks interface of the der operation station.

The “Remote Engineering” allows to run IndraWorks on a separate computer (control station), connected to the operation station via network.

To work in the Remote Engineering proceed as follows:

1. Enable the “Remote Engineering” on the operation station
2. On the projecting station:
   - Connect to the operation station
   - Open and edit project on operation station
   - Save the project
   - Disconnect from operation station

3. Now, you can disable the “Remote Engineering” on the operation station, if no more accesses of the projecting station are necessary.

2.14.2 Enabling and Disabling the “Remote Engineering” on the Operation Station

Enabling

On an IndraWorks operation station you must explicitly enable or disable the Remote Engineering.

You can do this via the “Remote Engineering Configuration” application in the start menu.

Fig.2-169: Remote Engineering Configuration, start

The dialog box of the “Remote Engineering Configuration” application displays the current state (here: disabled).

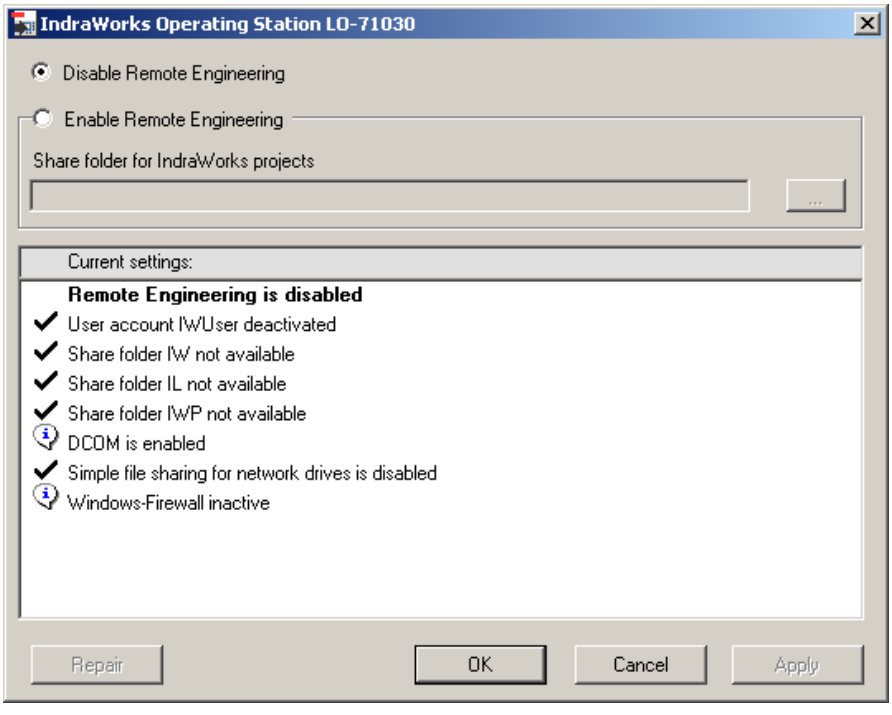


Fig.2-170: Remote Engineering disabled

Press “OK” or “Cancel” to close the dialog box. The remote engineering remains disabled.

Select “Enable Remote Engineering” and define the share folder for IndraWorks projects in an input box or selection dialog. Now, projection stations can access IndraWorks projects via this directory.

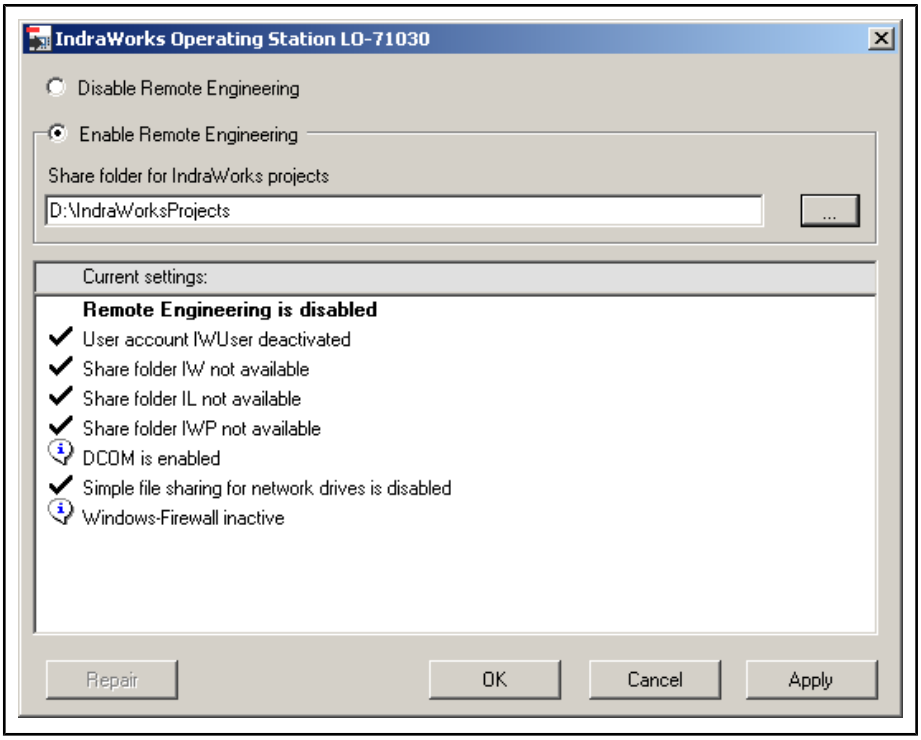


Fig.2-171: Define the share folder

The dialog box displays the current mode and the according settings. Valid settings have got a check mark. Invalid settings are marked by a white X on a red circle. Use the “Repair” button to correct the setting; this can take some time.

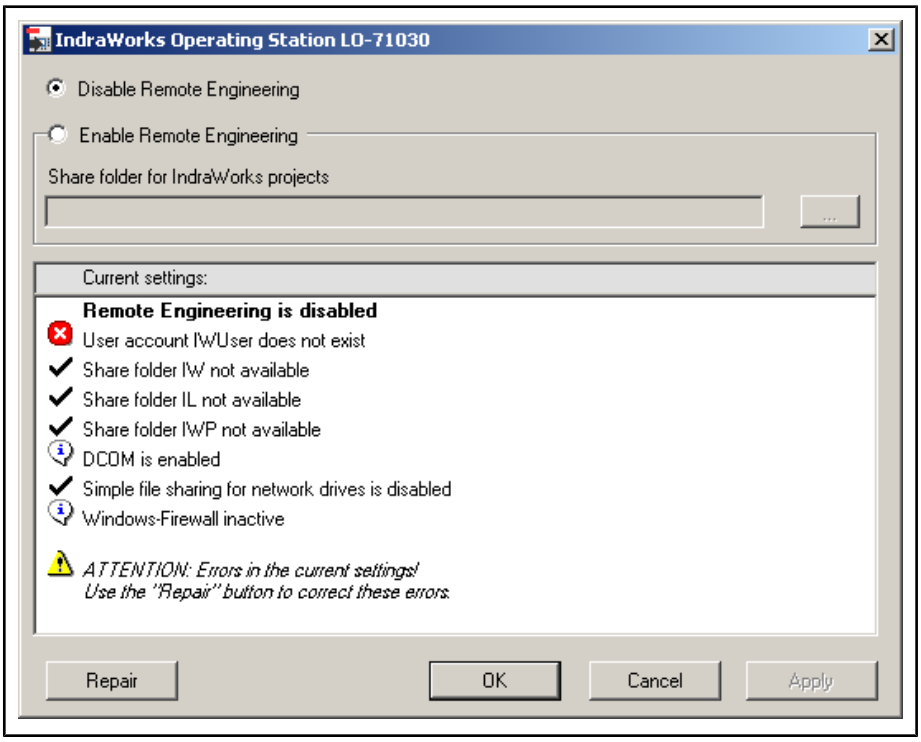


Fig.2-172: Errors in the current settings

“Cancel” closes the dialog box without any changes.

Use “OK” and “Apply” to start the following procedure:

- A user, internally used by IndraWorks, will be created or its account will be activated.
- The connected projecting station will get full access to the following directories, using the Windows security settings:
  - IndraWorks projects directory
  - IndraWorks installation directory
  - IndraLogic installation directory
- Share folders for these three directories will be created.

As long as the procedure is running, the dialog cannot be edited and the cursor appears as an hourglass.

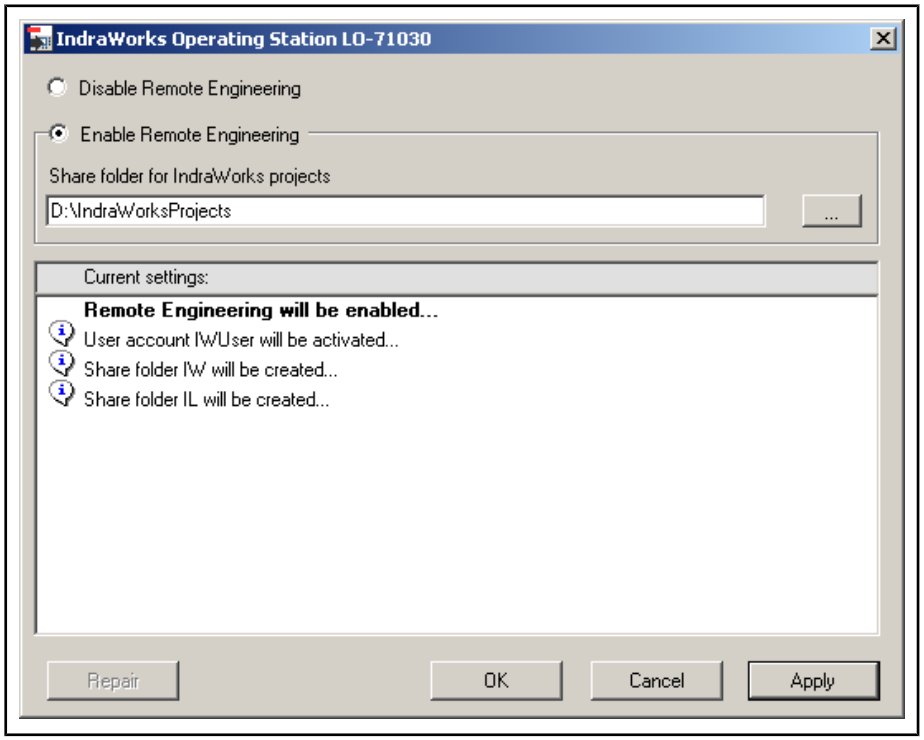


Fig.2-173: Remote Engineering will be enabled.

“OK” closes the application.

Disabling

If you want to disable the Remote Engineering, start the “Remote Engineering Configuration” via the start menu. This dialog box displays the current state (here: enabled).

Working with IndraWorks

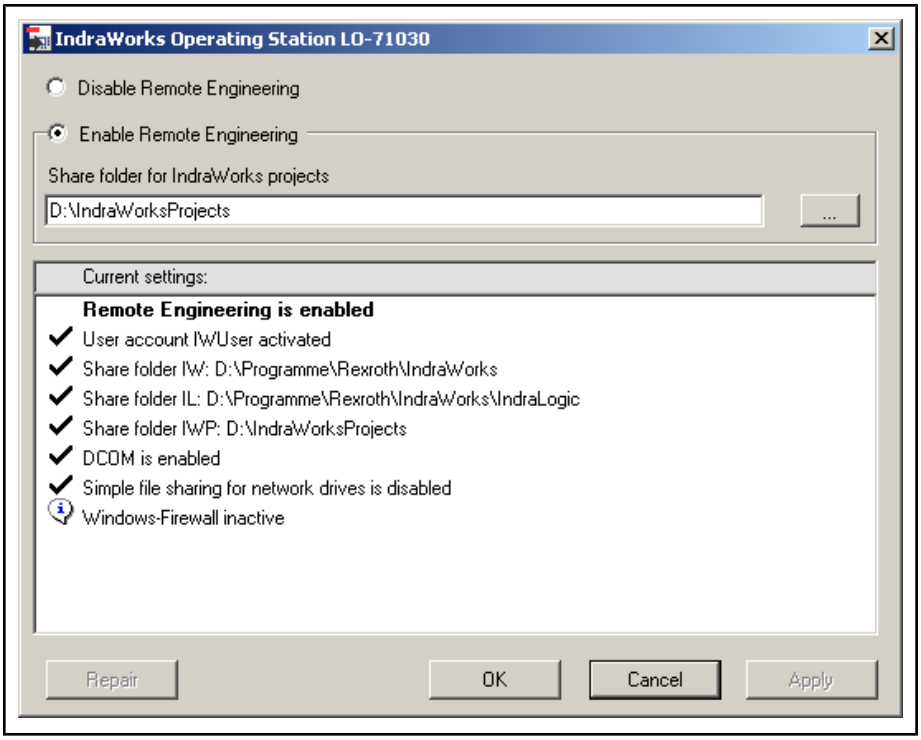


Fig.2-174: Enabled Remote Engineering

Disable the Remote Engineering by “Disable Remote Engineering” and “Apply” or “OK”. The steps described above will be performed in reverse order.

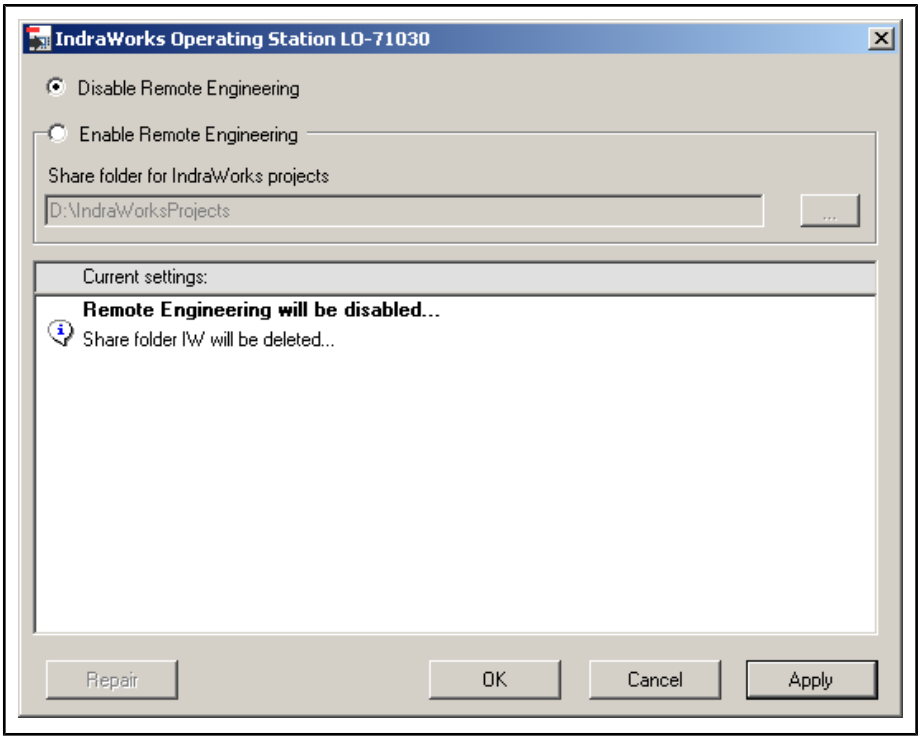


Fig.2-175: Disable Remote Engineering

“OK” stops the Remote Engineering.

2.14.3 Operations at the Projection Station

Connecting to the Operation Station

This function is available, only if you have installed the IndraWorks LOGIC system or the IndraMotion MTX system.

First the according operation station must be enabled for the Remote Engineering (see above).

To connect to the operation station select the according menu entry in IndraWorks.

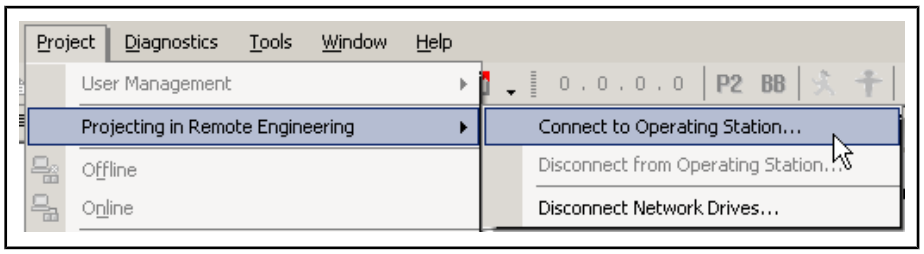


Fig.2-176: Connect to Operating Station, menu entry

If a project is open, it will be closed automatically after the confirmation of a message box.

Now you are asked to enter the host name or IP address of the operation station.

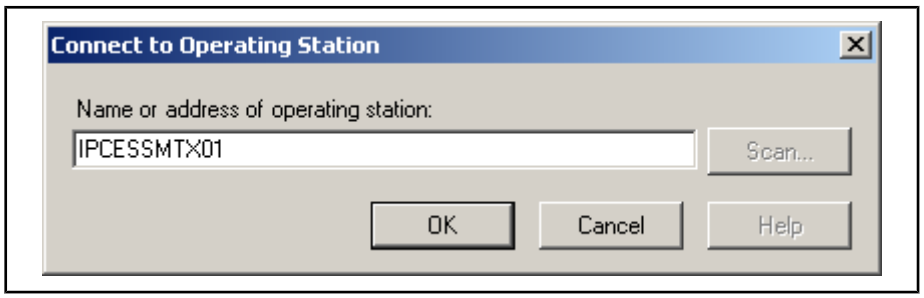


Fig.2-177: Connect to Operating Station dialog box, input of host name

“Cancel” closes the dialog box without any changes.

“OK” starts the following procedure:

- Three network connections to the specified host will be established.
  - \\<Host Name>\IW IndraLogic installation directory.
  - \\<Host Name>\IW IndraWorks installation directory.
  - \\<Host Name>\IWP IndraWorks project directory.
- The Remote Engineering will be enabled.

Fig.2-178: Release the connection

Remote Engineering

If Remote Engineering is enabled, the working with IndraWorks will be changed as follows:

- Three network drives are available to access directories of the connected operation station.

- The dialogs “Open Project”, “Create New IndraWorks Project” and “Save Project As” refer to the project directory of the operation station.
- Use **Project ▸ Active for IndraWorks Operation** to open the “Activate Project” dialog box and to define the project to be loaded for the operation interface of the connected operation station.
- The communication servers of the operation station will be used.

Disconnect from Operation Station

An existing connection to an operating station can be closed by **Project ▸ Remote Engineering ▸ Disconnect from Operating Station...**.

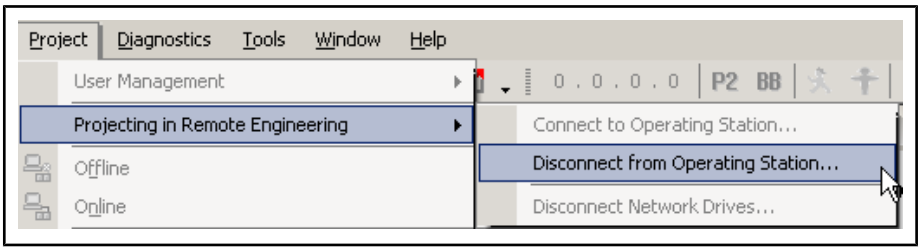


Fig.2-179: Disconnect from Operating Station, menu entry

If a project is open, you will be prompted to close it.

The disconnection from the operation station causes

- the network connections to be closed.
- the disconnecting from the communication servers of the operation station.

The state will be displayed in the status bar.

Fig.2-180: Connection closed

Disconnecting Existing Network Drives

You have to close all existing network connections to the operation station in order to connect to this operation station, because Windows does not allow to establish network connections to one host using different user names.

Open the **Disconnect Network Drives** Windows dialog box via the according menu entry and close all existing network connections to the operation station you want to connect to via the IndraWorks projection interface.

2.15 Remote Service

2.15.1 General Information

General With I-Remote, an efficient teleservice software is available in IndraWorks for the remote control of PC systems. The use of I-Remote allows you to access remote machines and to perform diagnostics or maintenance steps.

Software Components The I-Remote teleservice software consists of two software components.

Software Component	Description
Client	Software component installed on the PC to be remote-controlled. After completed installation, the client PC can be remote-controlled from a service PC.
Control	Software component installed on the remote-controlling PC. After completed installation, the control PC can remote-control PCs on which a client is installed.

Fig.2-181: I-Remote teleservice software

Delivery

The client installation is an integral part of IndraWorks and can be enabled by acquiring a valid license.

The control installation is delivered as a separate data carrier. Enabling of this component also requires a valid license. For detailed help on the installation and operation of the I-Remote control software, please refer to the data carrier of the control installation.

2.15.2 I-Remote Client Software

The sections following below refer exclusively to the I-Remote client software.

Installation

While IndraWorks is installed, the I-Remote setup for the client software is filed to each control PC.

To install the software, please proceed as follows:

- Select **Setup** in the **Start** ▸ **Programs** ▸ **Rexroth** ▸ **IndraWorks** ▸ **I-Remote** menu.
- Select the language and accept the license conditions.
- Enter the license information which you received when you purchased IndraWorks I-Remote in the “General” tab and click on “Generate”.

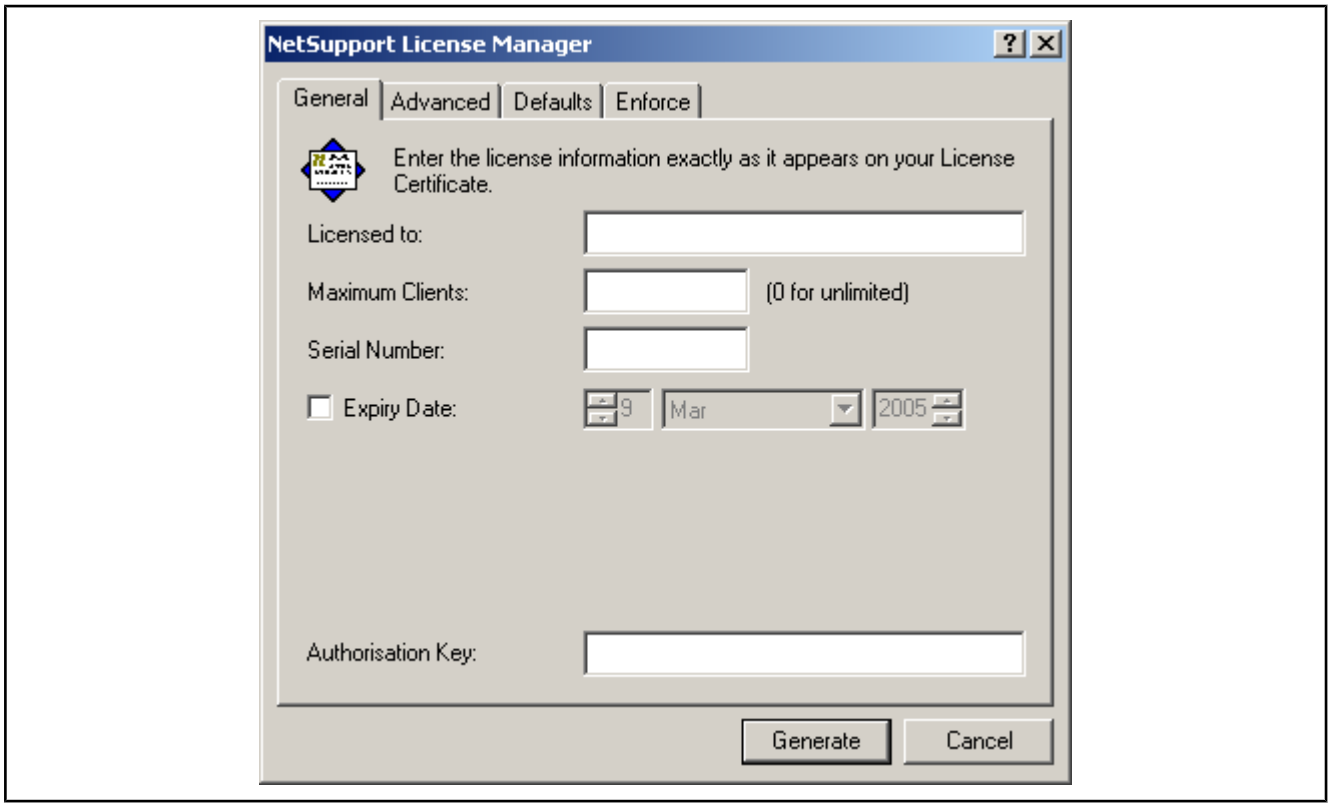


Fig.2-182: NetSupport License Manager dialog box

☞ Enter the information precisely, considering upper and lower cases.

- Click on “Install”.
- Close all applications and restart the computer. This completes the installation of IndraWorks I-Remote.

IndraWorks Integration

After successful installation, the **Tools** menu of the IndraWorks Engineering desktop contains **I-Remote** with the **Activate**, **Deactivate** and **Configuration** menu items.

Authorization

The use of the I-Remote functions requires the following authorizations:

- You must be logged in as main user or administrator in Windows.
- The user logged in IndraWorks must have the privilege “I-Remote” or he must be a member of a group with this privilege (see chapter 2.8 "User Management" on page 82).

Activate

This menu entry activates the I-Remote software. You can now access this PC from a service PC with the I-Remote control software. For safety reasons, the user must confirm a connection request of the service PC in the default configuration.

Deactivate

Deactivates the I-Remote software. The PC cannot be accessed via I-Remote any longer.

Configure

The I-Remote session between the service PC and the client PC can be configured as desired with regard to important configuration parameters, e. g.:

- Safety (user management, access rights)
- Transport protocol
- Functionality available during access
- WEB interface

While an I-Remote session is being established, the configuration is valid for both users of the session, i. e. for the service PC and for the client PC.

Please, use the supplied documentation and online help of the NetSupport Manager to make adjustments to the configuration.

2.16 External Applications

2.16.1 General Information

IndraWorks provides the option of calling any external applications and HTML pages. Once they have been configured appropriately, these programs and pages are displayed in the **Tools** menu and can be started by selecting the associated entry.

To configure and manage external applications, select **Tools ▸ External Applications**.

☞ When you are using **Automation Interface**, create an HTML page with embedded script code containing the access functionality to the Automation Interface. Add this functionality to the Tools menu as described in the following chapter.

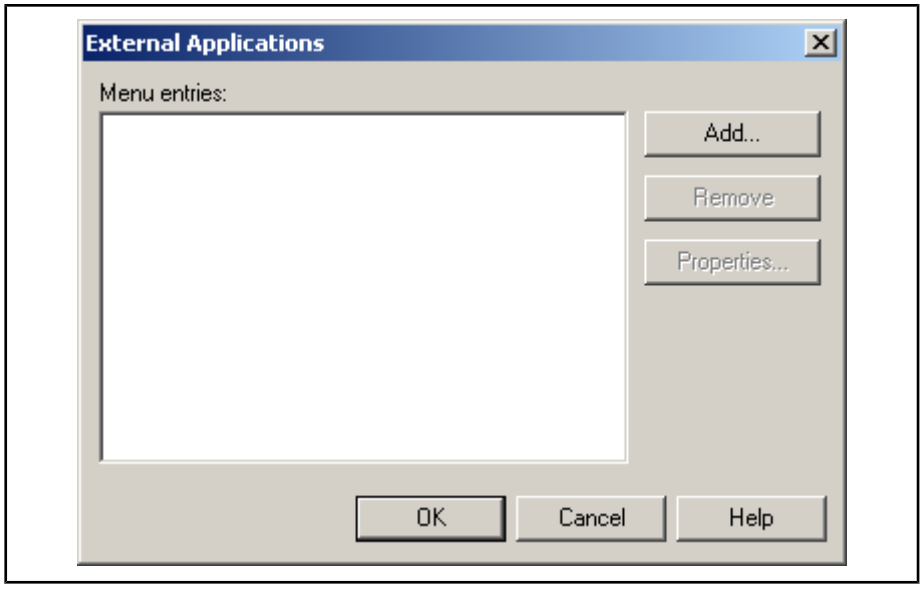


Fig.2-183: External Applications dialog box

2.16.2 Adding an Application or HTML Page

Use “Add” to open a dialog box and to select an application or HTML page. To select HTML pages enter "All Files" in the file type field.

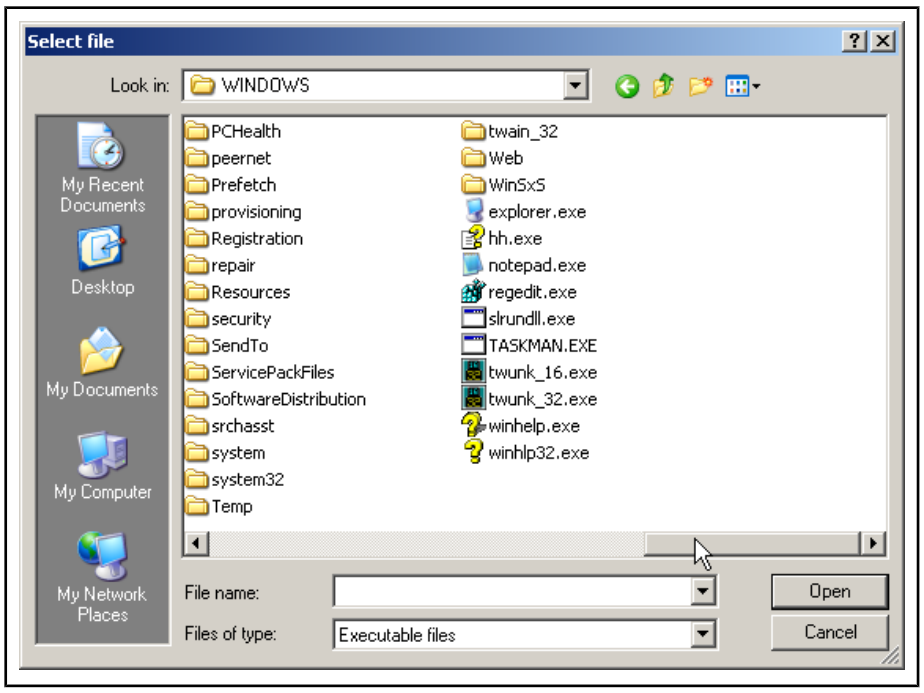


Fig.2-184: Select File dialog box

Confirm the file selected with “Open”. The application or HTML page will be displayed in the Properties dialog box . There you can define the properties (see chapter 2.16.3 "Changing the Properties of an Application " on page 132).

After you have confirmed all dialogs, the application or HTML page is displayed and can be called in **Tools ▸ External Applications...**.

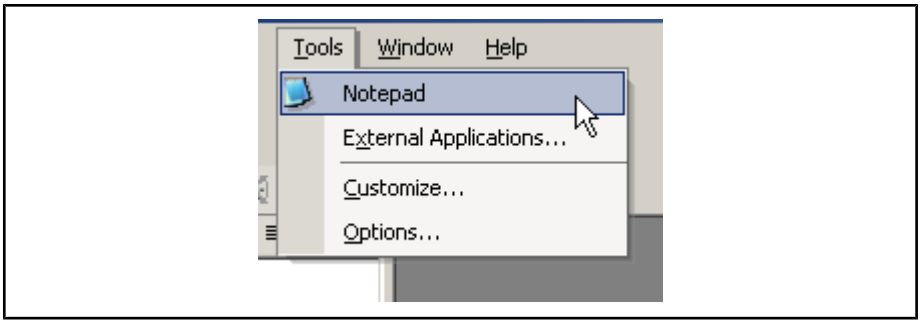


Fig.2-185: External application in the Tools menu

2.16.3 Changing the Properties of an Application

To open the Properties dialog box for an application, select it in the “External Applications” dialog box and click on “Properties”.

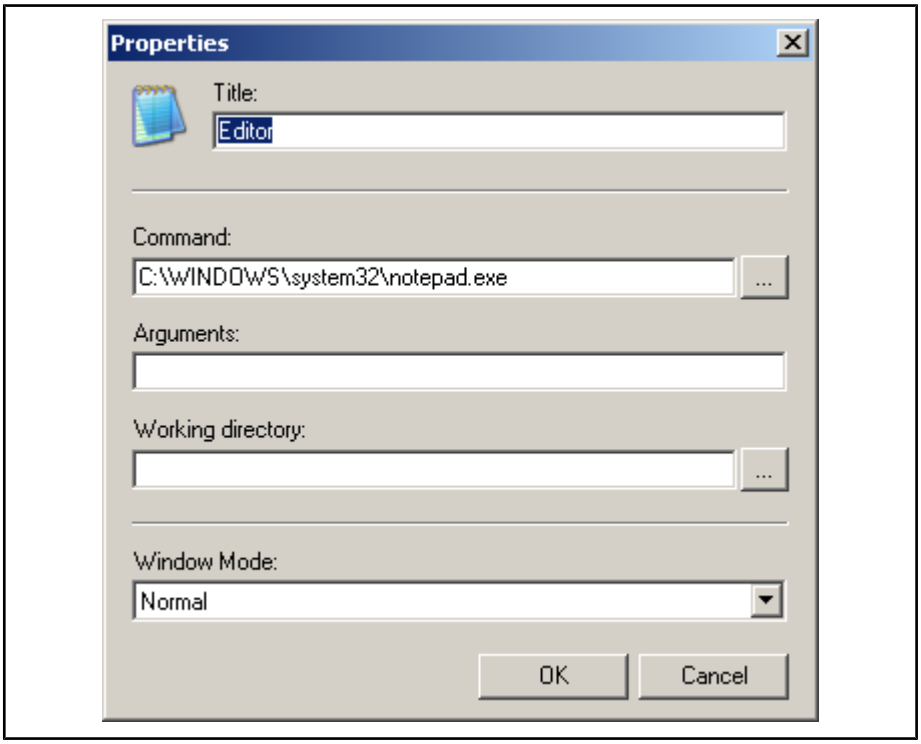


Fig.2-186: Properties dialog box, properties of HTML pages

Title This field automatically contains the descriptive text of the application. The title is also used in the menu as display text.

Command Enter the application file in this field or select it by via “...”.

Arguments Here you can optionally enter arguments to be transferred to the application when it is started.

Working Directory Optionally enter the working directory here or select it by clicking on “...”.

Window Mode Select here whether the application will be started wit normal window size or in full screen mode, as far as the application uses this parameter.

2.16.4 Changing the Properties of an HTML Page

To open the Properties dialog box for an HTML page, select it in the “External Applications” dialog box and click on “Properties”.

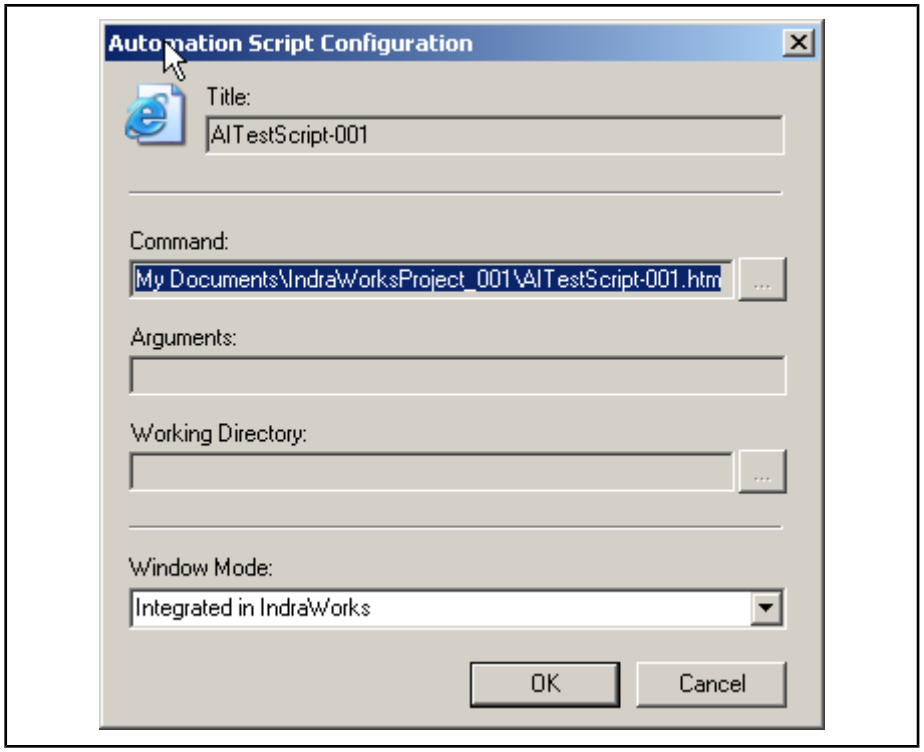


Fig.2-187: Properties dialog box, properties of HTML pages

Title This field contains automatically the descriptive text of the HTML page. The title is also used in the menu as display text.

Command Enter the file name of the HTML page in this field or select it via "...".

Arguments This field is locked when adding HTML pages.

Working Directory This field is locked when adding HTML pages.

Window Mode Here, you can define whether the HTML page is to be started integrated in IndraWorks or externally. The current version starts HTML pages only integrated in IndraWorks.

2.16.5 Removing an Application or HTML Page

To remove an application or HTML page from IndraWorks, select the appropriate entry from the "External Applications" dialog box and click on "Remove".

2.17 Options Dialog

2.17.1 General Information

The options dialog box provides comprehensive setting options allowing you to customize IndraWorks.

These setting options cover the appearance and the behavior of the user interface as well as specific settings of various program parts.

The setting options are organized on options pages. A single page contains coherent or related settings.

Select **Tools** ▸ **Options** to open to the Options dialog box.

2.17.2 Function Areas

The Options dialog box is divided in a navigation area and an area for displaying the options pages.

Navigation Area The navigation area contains the tree-like hierarchy of the options pages. The highest level only comprises folders. Folders can contain options pages and subfolders.

Options Pages Select an element from the navigation area to display the appropriate options page. Here you can view and edit the particular settings.

Select a folder from the navigation area to open an overview of its available options pages and a brief description.

2.17.3 Operation

Changing Settings Select an element from the navigation area of the Options dialog box to display the corresponding options page.

Make the desired settings on that options page. Several options pages can be edited one after the other. These changes are preserved as long as the Options dialog box is open.

Making Changed Settings Effective Changes will become effective only after you have clicked on “Apply” or “OK”.

Restoring Settings to Default Values To restore the default values on the current options page, click on “Reset”. Then click on “Apply” or “OK” so that the default settings will become effective.

Using Help Click on “Help” to open the online help for the current options page.

Exiting the Options Dialog Box Click “OK” to exit the Options dialog box. The changes will now become effective.

To discard all changes and to close the Options dialog box, click on “Cancel”.

2.17.4 General Options

Language Settings

Use the “Language settings” options page to select the user interface language for IndraWorks.

As soon as the language selected has become effective by clicking on “OK” or “Apply”, the display of the user interface is updated, i. e. the user interface is displayed in the language desired immediately.

Software Licenses

Use this page to manage the licenses of software components. (see chapter 2.12 "License Management " on page 112)

Keyboard (Shortcuts)

Use the “Keyboard” options page to assign shortcuts to commands.

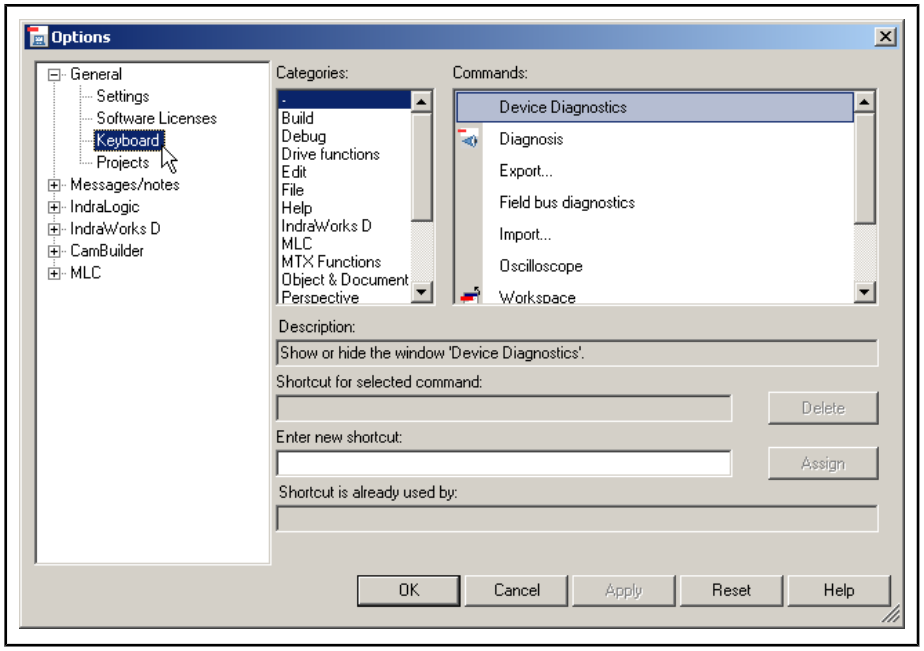


Fig.2-188: Options dialog box, keyboard

Categories Select a category to list the commands of that category in the “Commands” column.

Description This field provides a brief description of the command selected.

Shortcut for Selected Command If assigned to a selected command, a shortcut is shown in this field. Click on “Delete” to remove the shortcut. If a shortcut does not exist, this field shows “None”.

Enter New Shortcut Use this field to define a new shortcut for the command selected by pressing a key or a combination of keys. If the shortcut is already assigned to a different command, this is displayed in the “Shortcut is already used by” field.

A shortcut can be assigned to one command only. If you assign an already used shortcut to a new command, that shortcut cannot be used for the previous command any longer.

Projects

Use this options page to change the settings for the project explorer.

These changes will not become effective before the next restart.

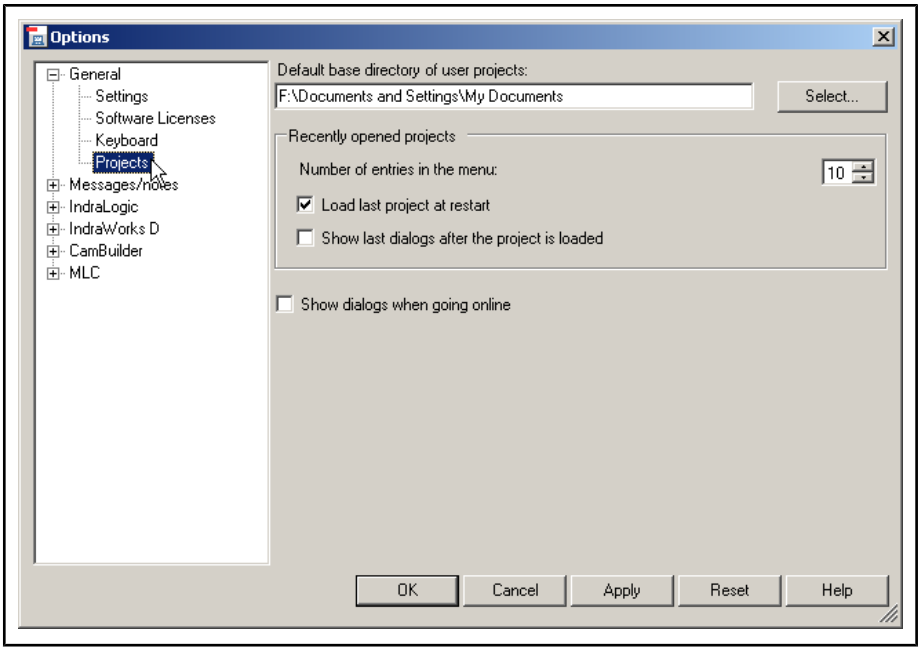


Fig.2-189: Options dialog box, Project explorer

2.18 Customizing Dialog

The “Customizing” dialog box provides the option of editing the menu and the toolbars.

To open this dialog box, select **Tools ▸ Customizing…**.

Adding a Command to the Menu or to a Toolbar

To add a command to a toolbar, activate that toolbar. Afterwards, select **Tools ▸ Customizing…** and the “Commands” page. Select the category for the command. Drag the command to be added from the “Commands” field to the appropriate point in the menu or the toolbar.

Showing and Hiding Toolbars

Toolbars can be shown or hidden. To do this, select **Tools ▸ Customizing…** and the “Toolbars” page. Make the appropriate settings in the “Toolbars” field.

Creating a New Toolbar

To create your own toolbar, click the “New…” button on the “Toolbars” page. This will open the “Create New Toolbar” dialog box where you enter a name for the new toolbar. Click “OK” to create the toolbar, which will be floating. The toolbar name will be entered in the toolbar list.

Renaming a Toolbar

Select the user-defined toolbar to be renamed from the “Toolbars” field on the “Toolbars” page. Click on “Rename”.

☞ Only user-defined toolbars can be renamed.

Deleting a Toolbar

Select the user-defined toolbar to be deleted from the “Toolbars” field on the “Toolbars” page. Click on “Delete” and confirm the “Delete Toolbar” message.

☞ Only user-defined toolbars can be deleted.

2.19 Info

Use **Help ▸ Info** to call the information dialog box of IndraWorks. This dialog box provides detailed information on the various components. The upper part of the dialog box shows the precise version and the build number of the basic installation. The lower part of the dialog box displays the license conditions.

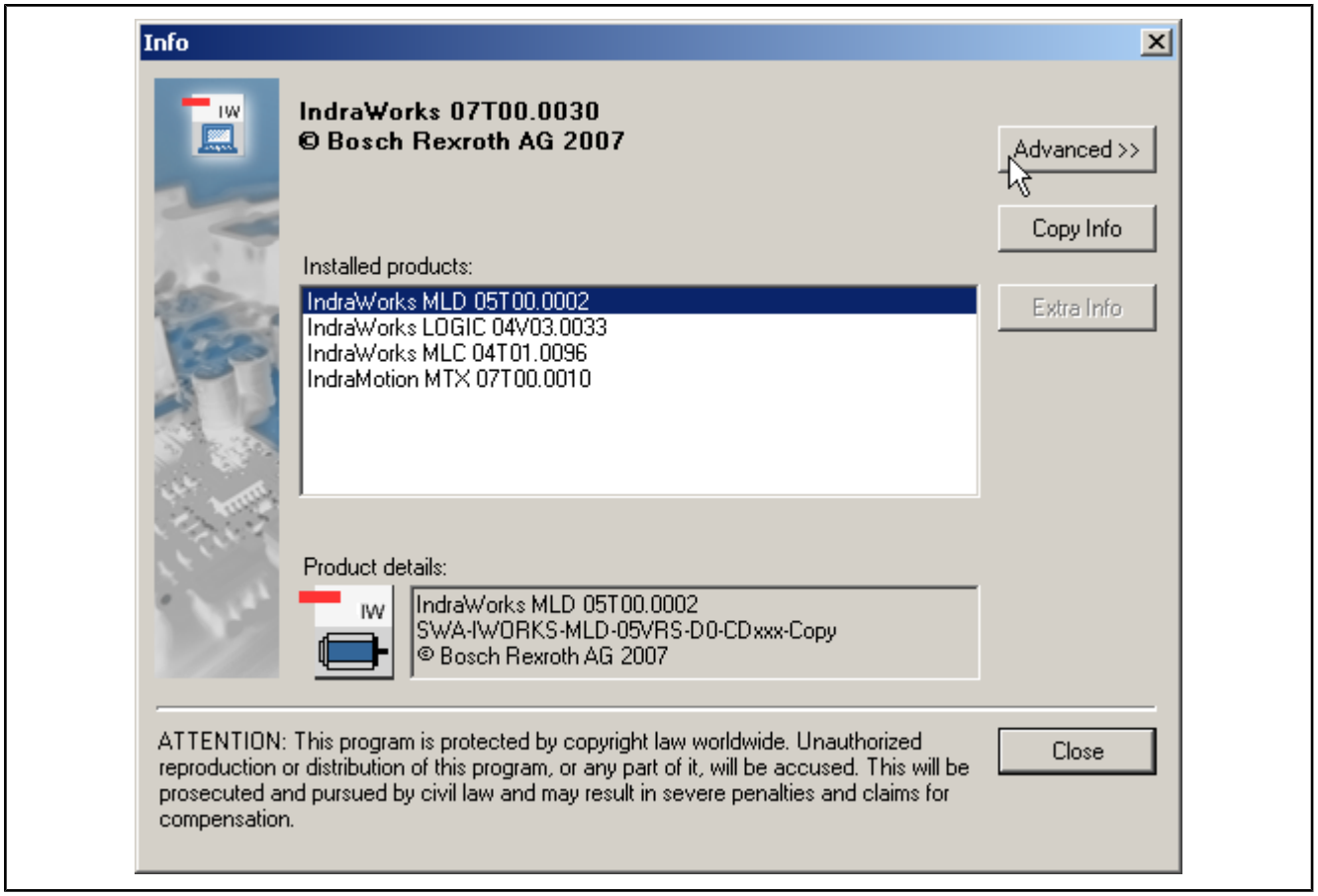


Fig.2-190: Info dialog box

Installed Products This list displays the products installed.

> You can display the installed products only, or the installed products together with their basic components (see Advanced Information).

Product Details Select a product from the list to view detailed information (type code and copyright) in the “Product details” field.

Advanced Information Click on the “Advanced>>” button to show the basic components of the installed software packages in the “Installed products” list. To hide this information again, click on “Advanced<<”.

Copy Info Click on “Copy Info” to copy the information displayed. IndraWorks copies not only the build and version information on IndraWorks itself, but also the information on assemblies, such as file size, creation date and the important information on the particular computer, processor and operating system. After the copying is completed, an editor is started for processing that information .

Calling Extra Info To obtain extra info on the product selected, click on the “Extra Info” button.

2.20 Message Box

IndraWorks provides a mechanism for recording errors and messages.

The message box is divided in three tabs each of which assumes one single function. In the normal operating mode, only two tabs are visible. The third tab is intended for diagnostic purposes during startup and can only be activated by an additional start parameter.

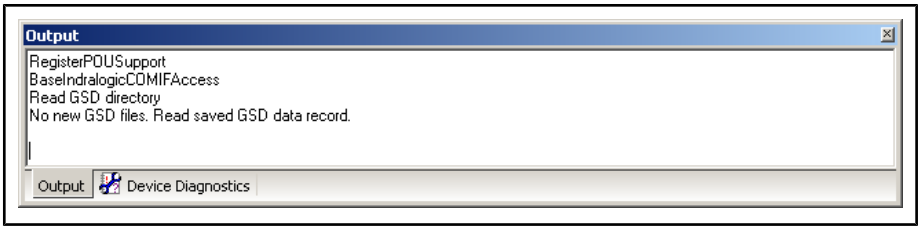


Fig.2-191: Output window

Output The “Output” window records processes such as a download or the compilation of a program.

Task List The “Task list” window lists failures and errors necessitating a reaction. If, for example, program errors are recorded here, these errors must be eliminated before the compilation process can be repeated without any errors. The window also displays the level and, thus, the error class and the origin of the respective error message.

Parameterizing the Message Box Use the “Logging Manager” page of the Options dialog box to parameterize the message box in IndraWorks.

Device Diagnostics You can show and hide the device diagnostics via **Diagnostics ▸ Device Diagnostics**. The device diagnostics shows a list of the current diagnostics of a device if

- this device or one of its sub-items is selected in the project explorer,
- the device supports diagnostic messages, and
- the device is in the “online” mode.

In all other cases, the list is empty.

Each diagnostic message shows the diagnostic number, the diagnostic source (module in case of ProVi messages, channel number in case of MTX), the date and time of occurrence as well as a descriptive text.

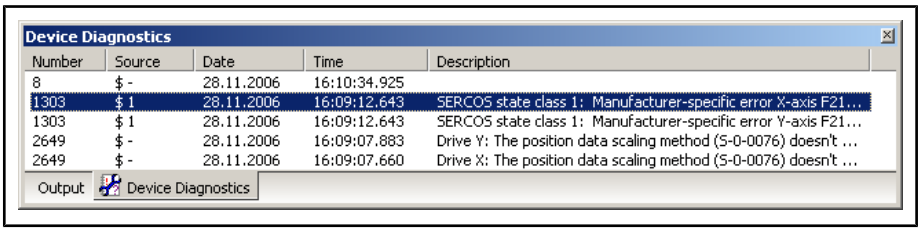


Fig.2-192: Device Diagnostics window

2.21 Help

2.21.1 IndraWorks Online Help

Help Use the **Help ▸ First Steps** command to start the online user documentation.

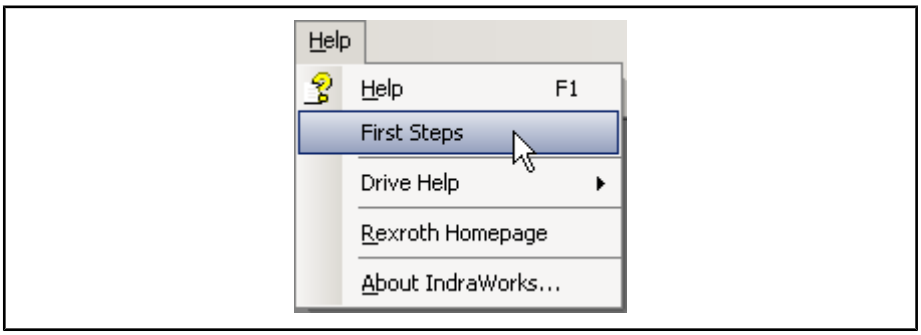


Fig.2-193: Help menu, first steps

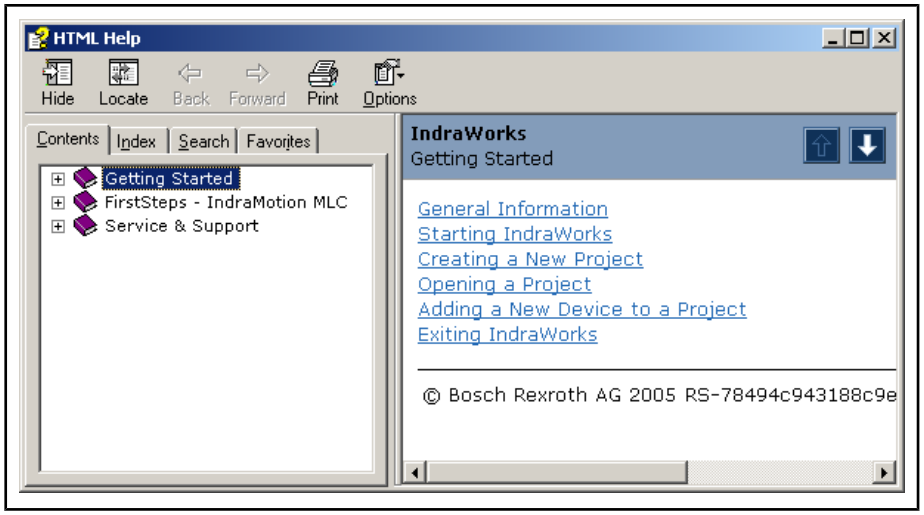


Fig.2-194: Online information on first steps with IndraWorks

Select **Help** ▸ **Help** to start the online user documentation of IndraWorks.

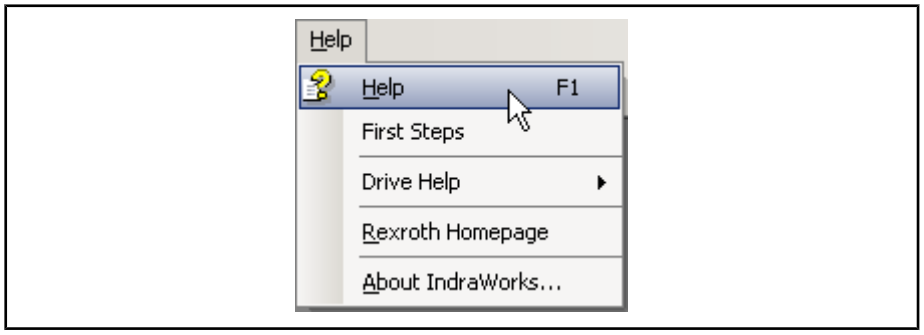


Fig.2-195: Help menu, start online help

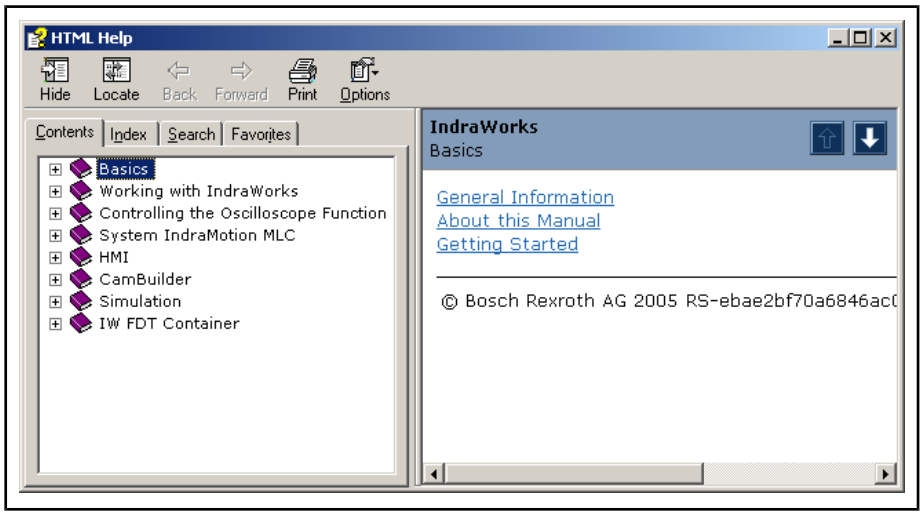


Fig.2-196: Online information on IndraWorks

Content You can reach all topics of the IndraWorks online help, sorted by headings, via “Content”.

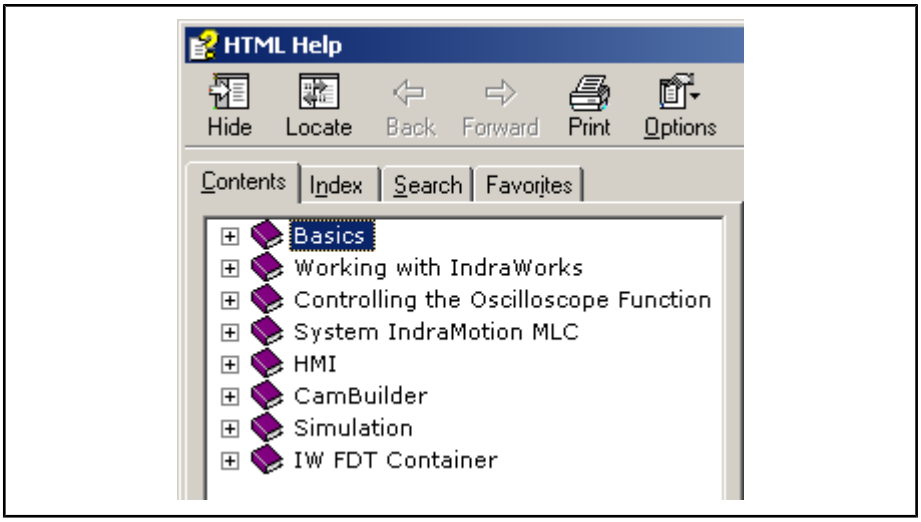


Fig.2-197: Contents of the IndraWorks online help

Index You can reach all topics of the IndraWorks online help, sorted by keywords, via “Index”.

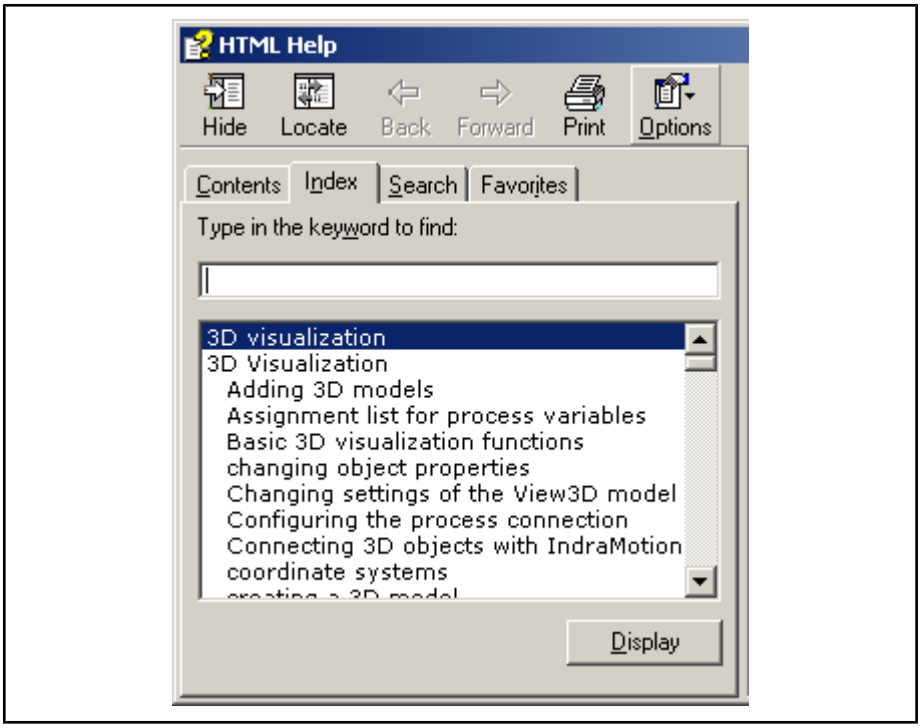


Fig.2-198: Help, Index

Search You can scan the IndraWorks online help for keywords via “Search”. The results are displayed in a list containing all associated topics.

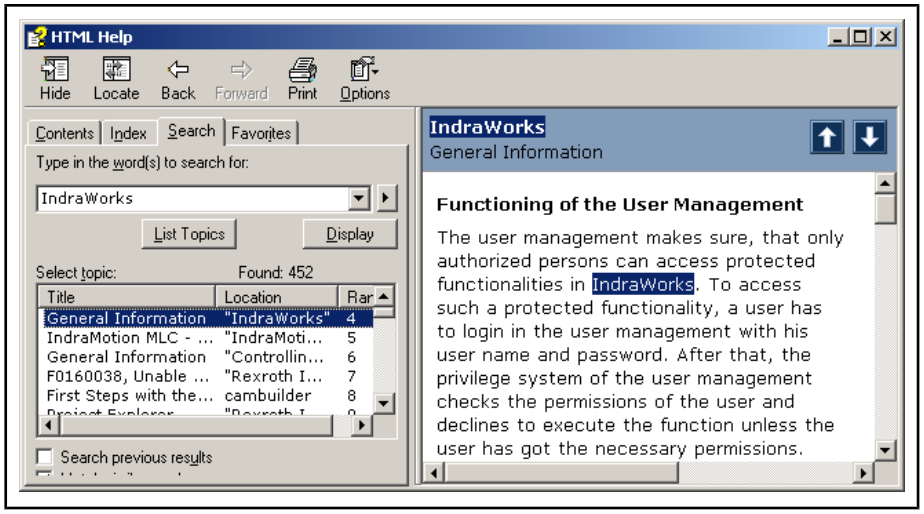


Fig.2-199: Help, search for topics

Rexroth Homepage

Select **Help ▸ Rexroth Homepage** to move to the Bosch Rexroth homepage, provided you are connected to the network.

3 Operation of the Oscilloscope Function

3.1 General Information

The oscilloscope function is used to display and evaluate measured values; that result, e. g., from the transient response of a motor.

Its main field of application is for commissioning of drives and controls (MTX), for test and service.

Requirements The device connected must support the oscilloscope function.

3.2 Starting the Oscilloscope Function

To activate the oscilloscope function select **Diagnostics ▸ Oscilloscope...** in the main menu.

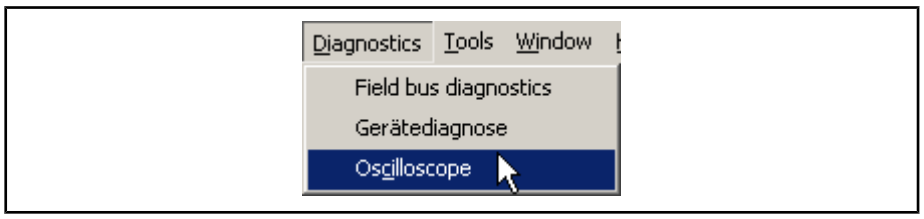


Fig.3-1: Start the oscilloscope

Offline Mode If no device is selected or if the online mode is not activated, the oscilloscope starts in the offline mode. This mode does not allow any measurements. However, already saved measurements resulting from other measuring processes can be read and displayed.

Online Mode If a device is selected and the online mode is activated, the oscilloscope starts in the online mode.

☞ You can also switch to the online mode subsequently.

3.3 Online and Offline Modes

3.3.1 Starting in the Offline Mode

If you start the oscilloscope in the offline mode, you cannot perform a measurement.

State The “State” field in the “Active measurement” area indicates that you are not connected to a device.

Operation of the Oscilloscope Function

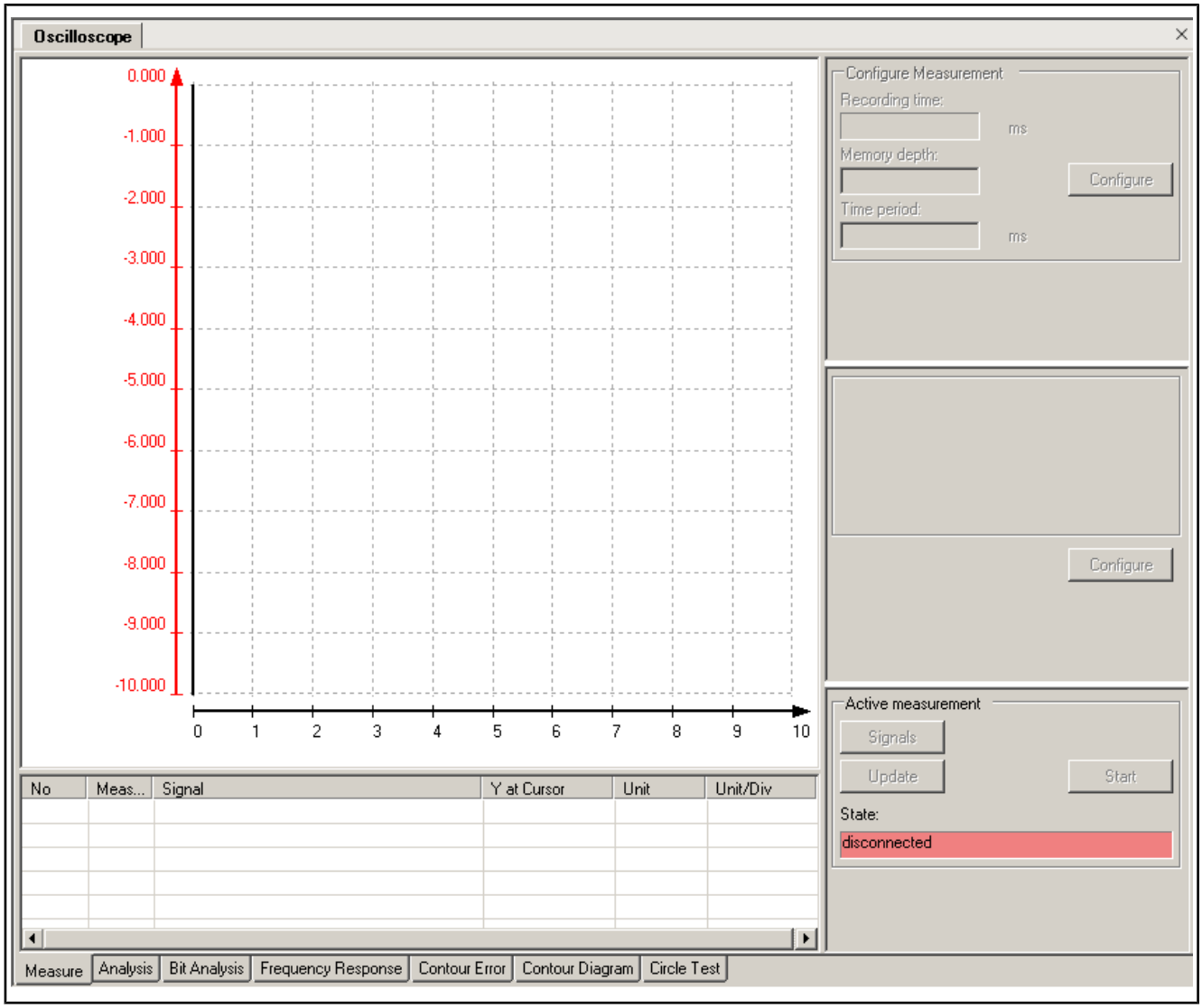


Fig.3-2: Oscilloscope, started in the offline mode

☞ In the offline mode you can read and analyze any measurements via the **Load Signal Data** or **Add Signal Data** menu items.

After having connected the project and the oscilloscope to a device, you can select the signals for the measurement. After that, you can configure the trigger and the measurement.

3.3.2 Starting in the Online Mode / Switching to the Online Mode

Active Measurement

After the switch to the online mode the oscilloscope reads the configuration of the device connected and displays an empty graph.

If the device contains a measurement, you can display it in the graph by clicking on the “Read Signal Data” button.

☞ The message displayed is called the current or active measurement.

A measurement contains the measured data and the configuration data.

State

The state of the measurement is displayed in the “State” field in the “Active measurement” area. If a measurement can be performed, its state is “ready”. Further states are “disconnected”, “not configured”, “not triggered”, “triggered” and “storing”.

Furthermore, the state field informs you on the progress of the reading process of the measurement data after the measurement. See the following example:

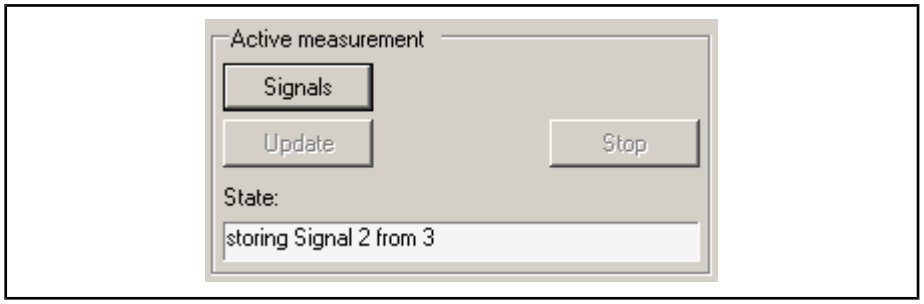


Fig.3-3: Progress when reading the measurement data

Signal overview

The signal overview shows the selected signals of the last measurement.

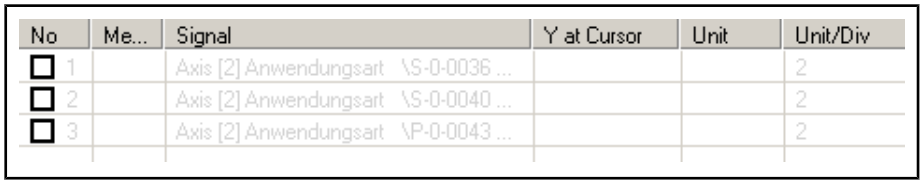


Fig.3-4: Display of the last configuration

If there are no signal data, you have to configure and start a measurement, see chapter 3.5 "Recording a New Measurement " on page 153.

Update

If the device contains not only the configuration but also signal data, e. g. after a long-term measurement, you can display them via “Update”. See the following example:

Operation of the Oscilloscope Function

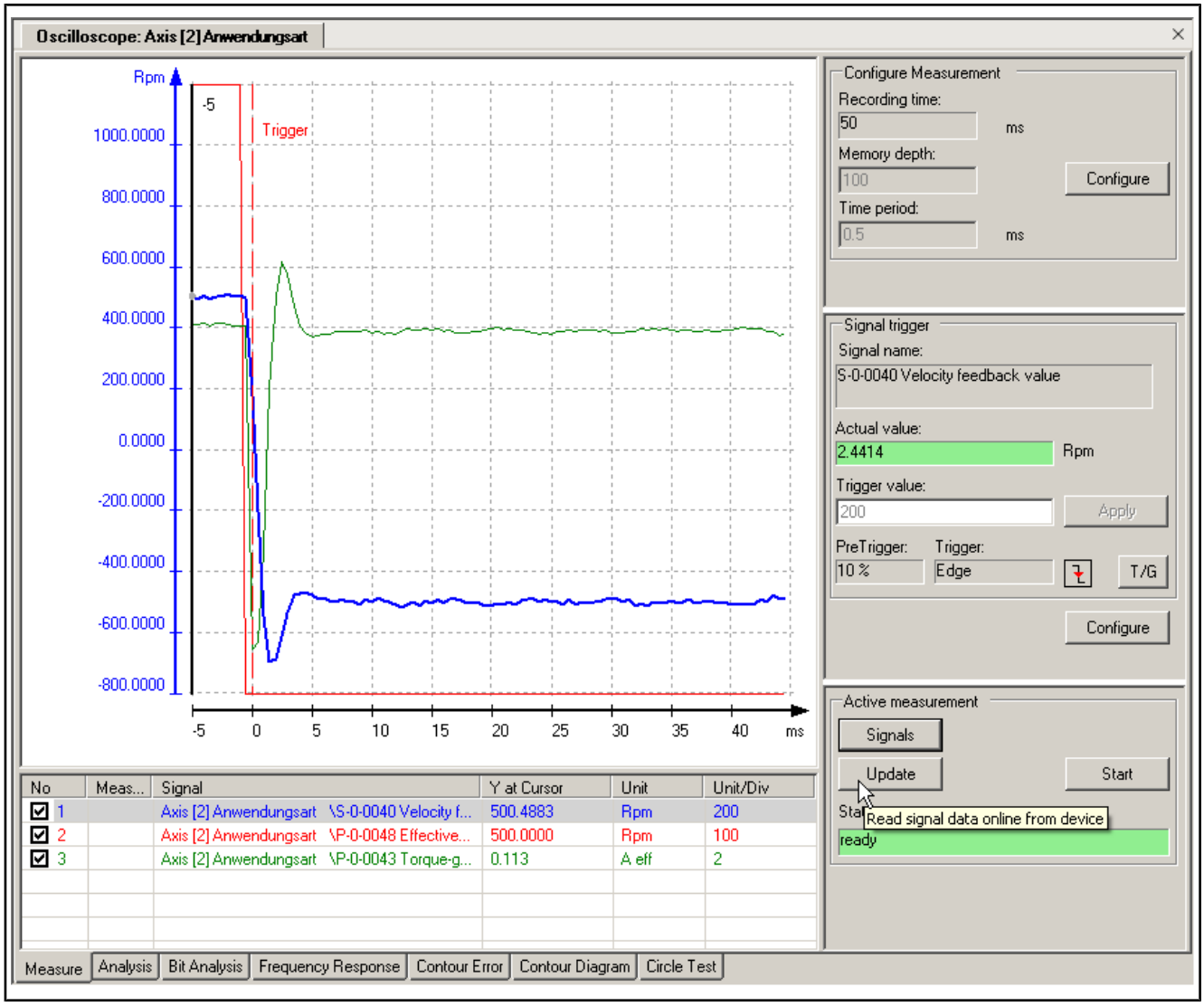


Fig.3-5: Display measurement data

To show or hide the separate signals set or delete the according checkmark.

Active Signal The display is always scaled according to the active signal. Active signals have a gray background in the signal overview and a greater width in the graph. To activate a signal, click on it in the graph or in the signal overview.

3.4 Operation Areas

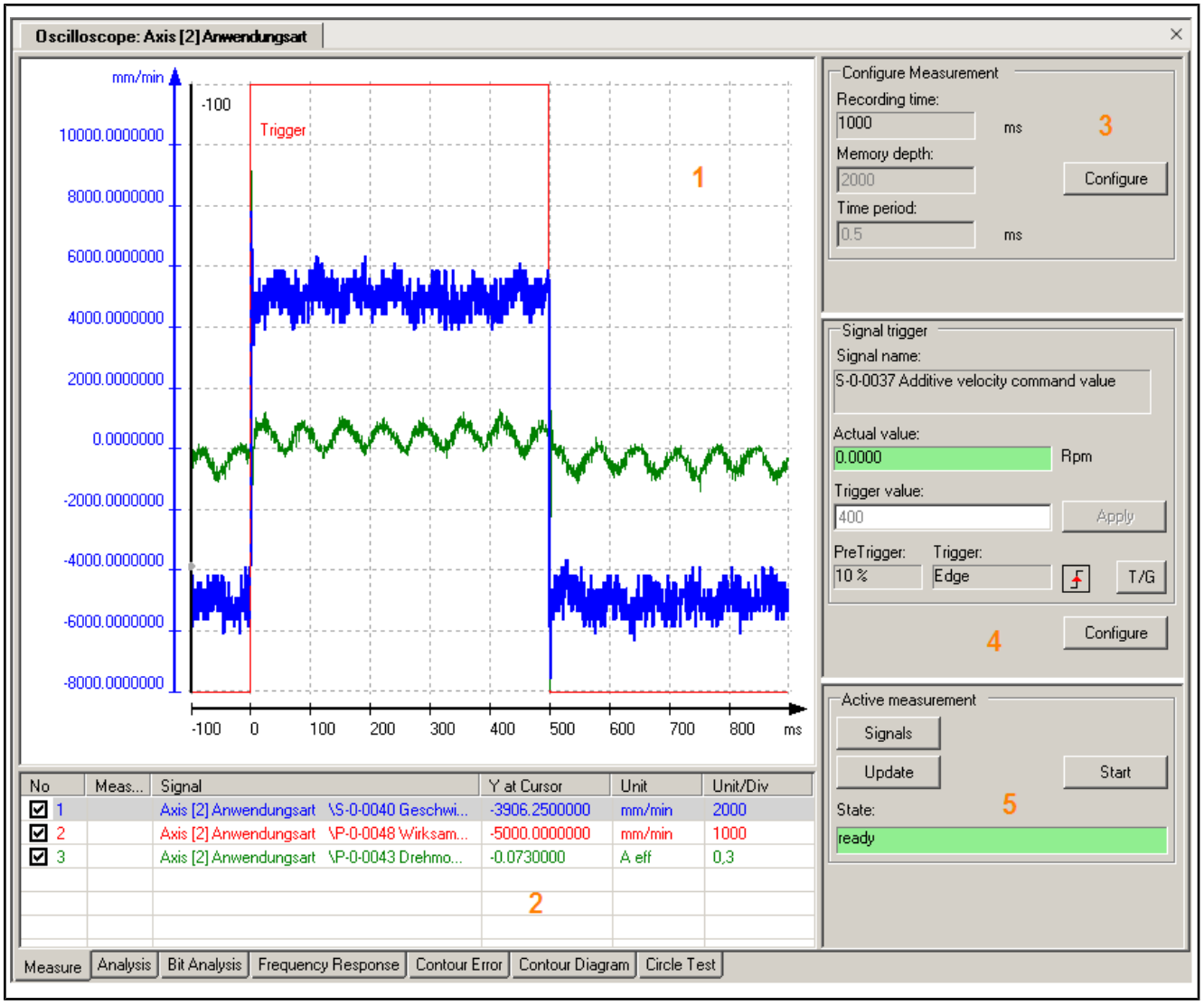
3.4.1 General Information

For a better clearness, the operation tasks of the oscilloscope are arranged on several tab pages.

Every tab page represents a specific topic.

3.4.2 Measure

Select the configuration or operation of a measurement on this tab page:



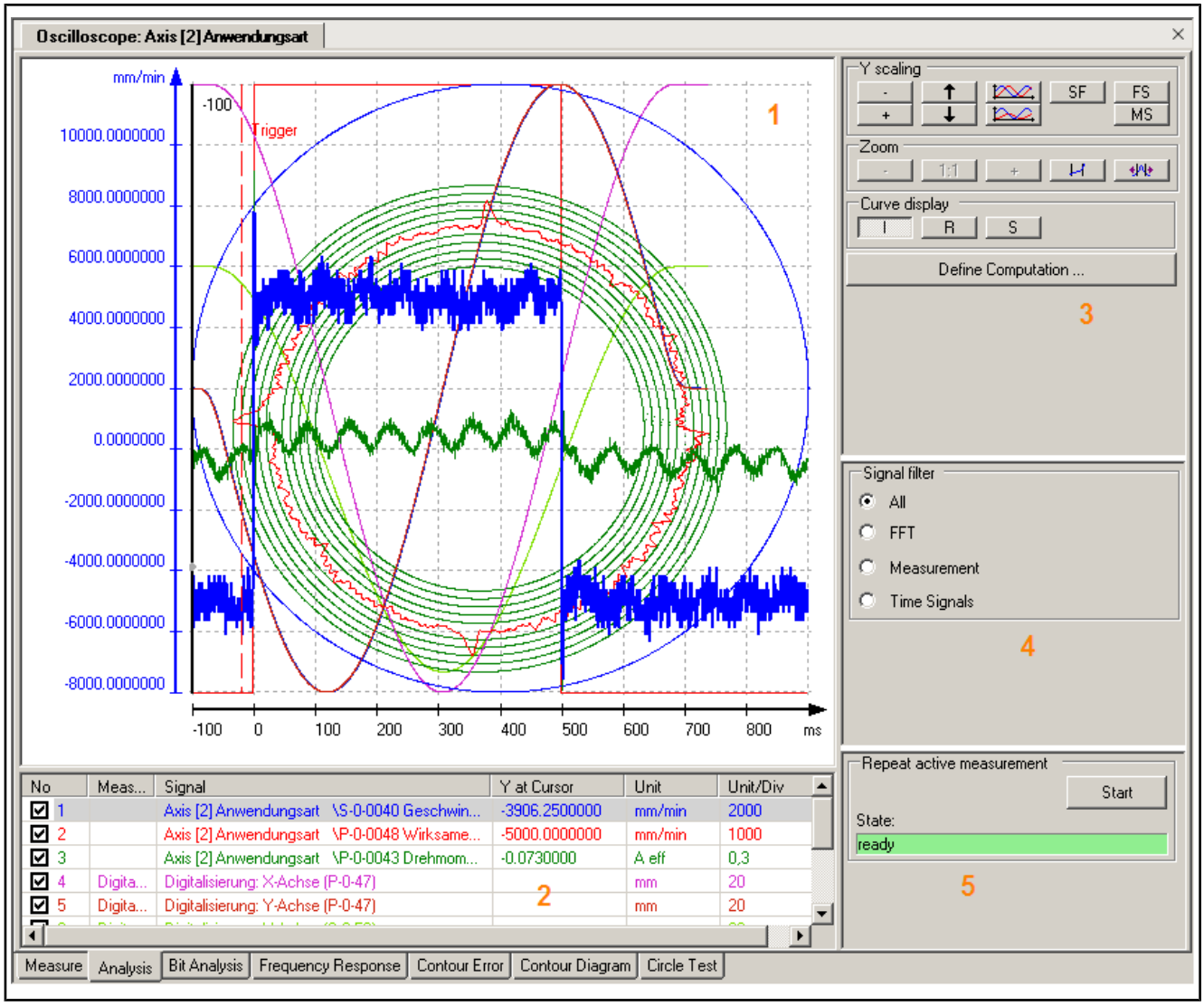
1	Graphical result
2	Measured signals
3	Display and modification of measurement configuration
4	Display and modification of trigger configuration
5	Selection of measurement signals, start and state of measurement

Fig.3-6: Configuration and measurement

3.4.3 Analysis

Use this tab page to adjust the graphical display:

Operation of the Oscilloscope Function

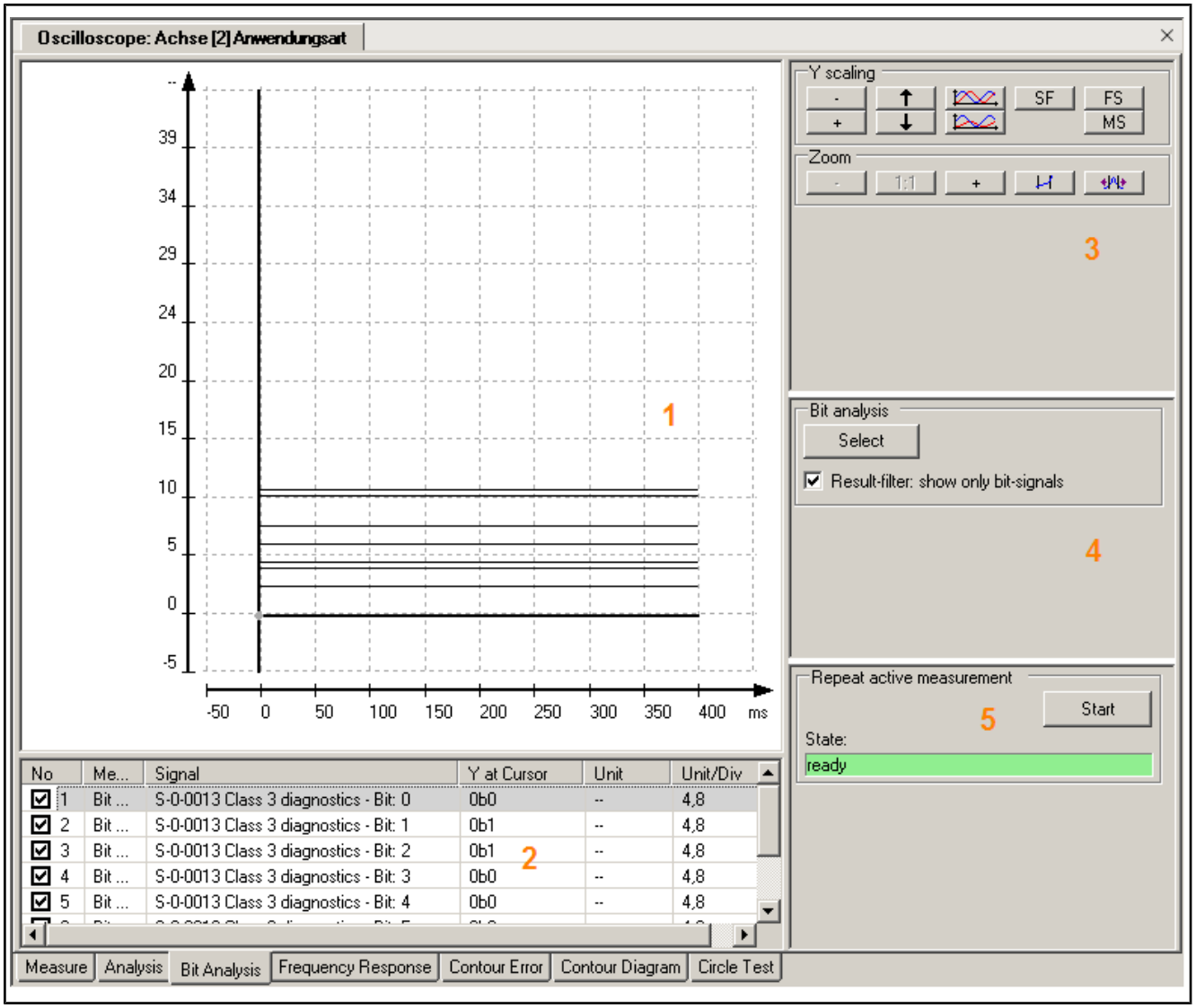


1 Graphical result
2 Measured and computed signals
3 Selection of Y scaling, zoom, curve display and computation
4 Switch signal filter on or off
5 Repeat measurement and state display

Fig.3-7: Analysis of measurement results

3.4.4 Bit Analysis

Use this tab page to display single bits of a signal, if possible.



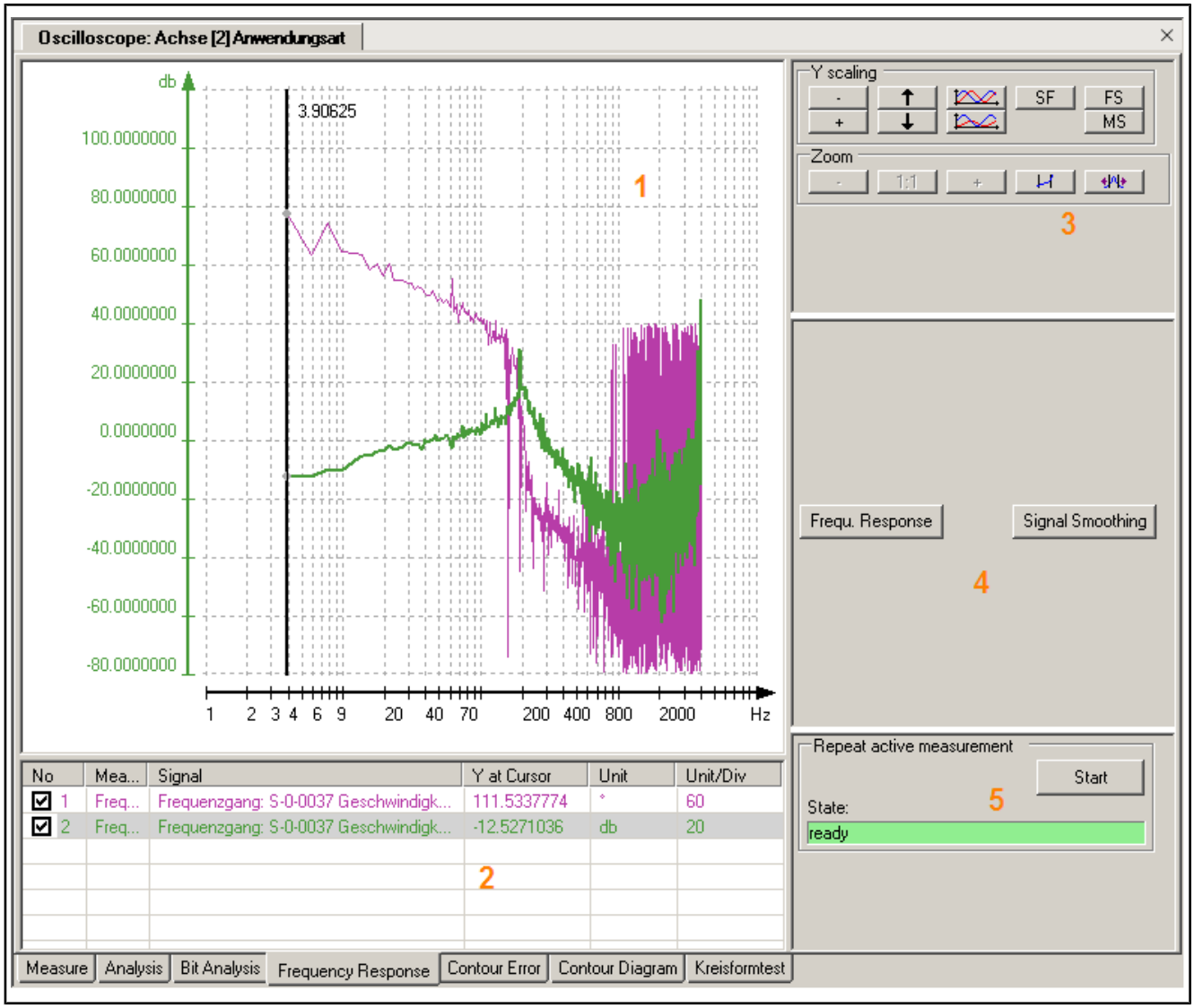
1 Graphical result
2 Computed signals
3 Y scaling and zoom
4 Bit selection via signal filter
5 Repeat measurement and state display

Fig.3-8: Extracting single bits

3.4.5 Frequency Response

To compute the frequency response after the measurement, select this tab page:

Operation of the Oscilloscope Function

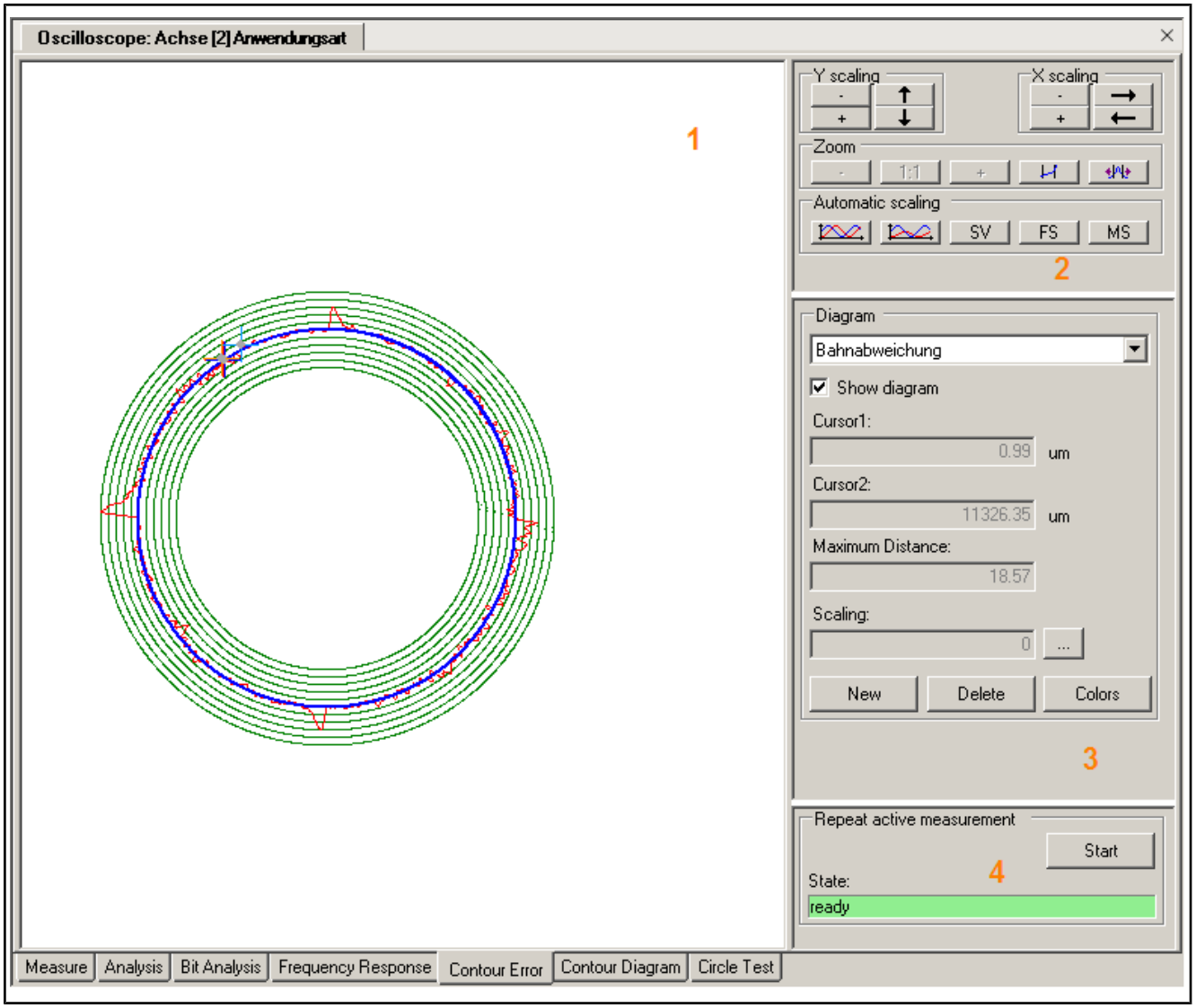


1	Graphical result
2	Computed signals
3	Y scaling and zoom
4	Selection of signals for computation and smoothing the result
5	Repeat measurement and state display

Fig.3-9: Computing the frequency response after the measurement

3.4.6 Contour Error

This tab page displays the contour deviation between command value and actual value after a measurement:



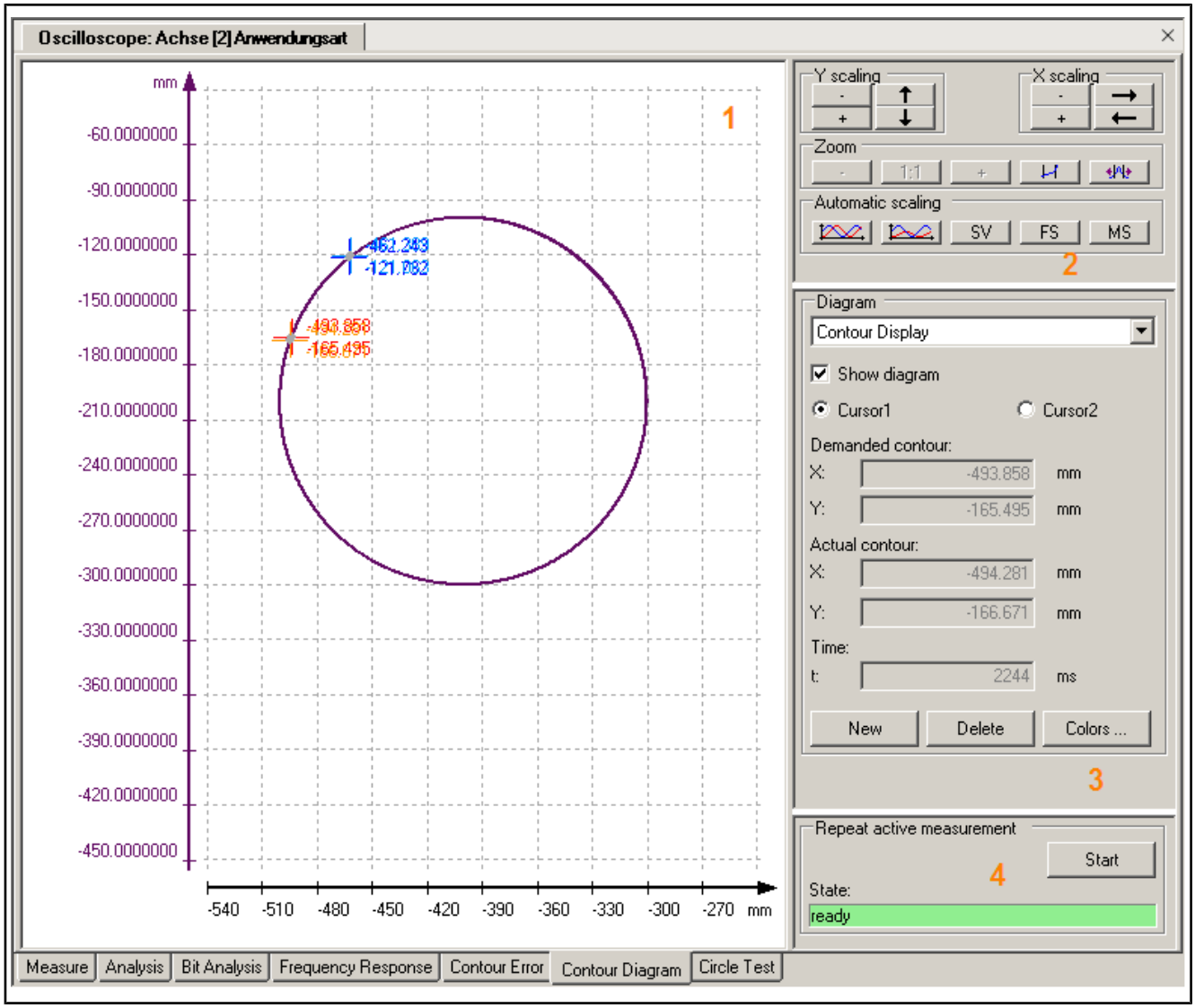
1	Graphical result
2	X or Y scaling and zoom
3	Operation with signal selection and cursor display
4	Repeat measurement and state display

Fig.3-10: Display of contour deviation

3.4.7 Contour Diagram

Select this tab page, to view a contour diagram after the measurement:

Operation of the Oscilloscope Function

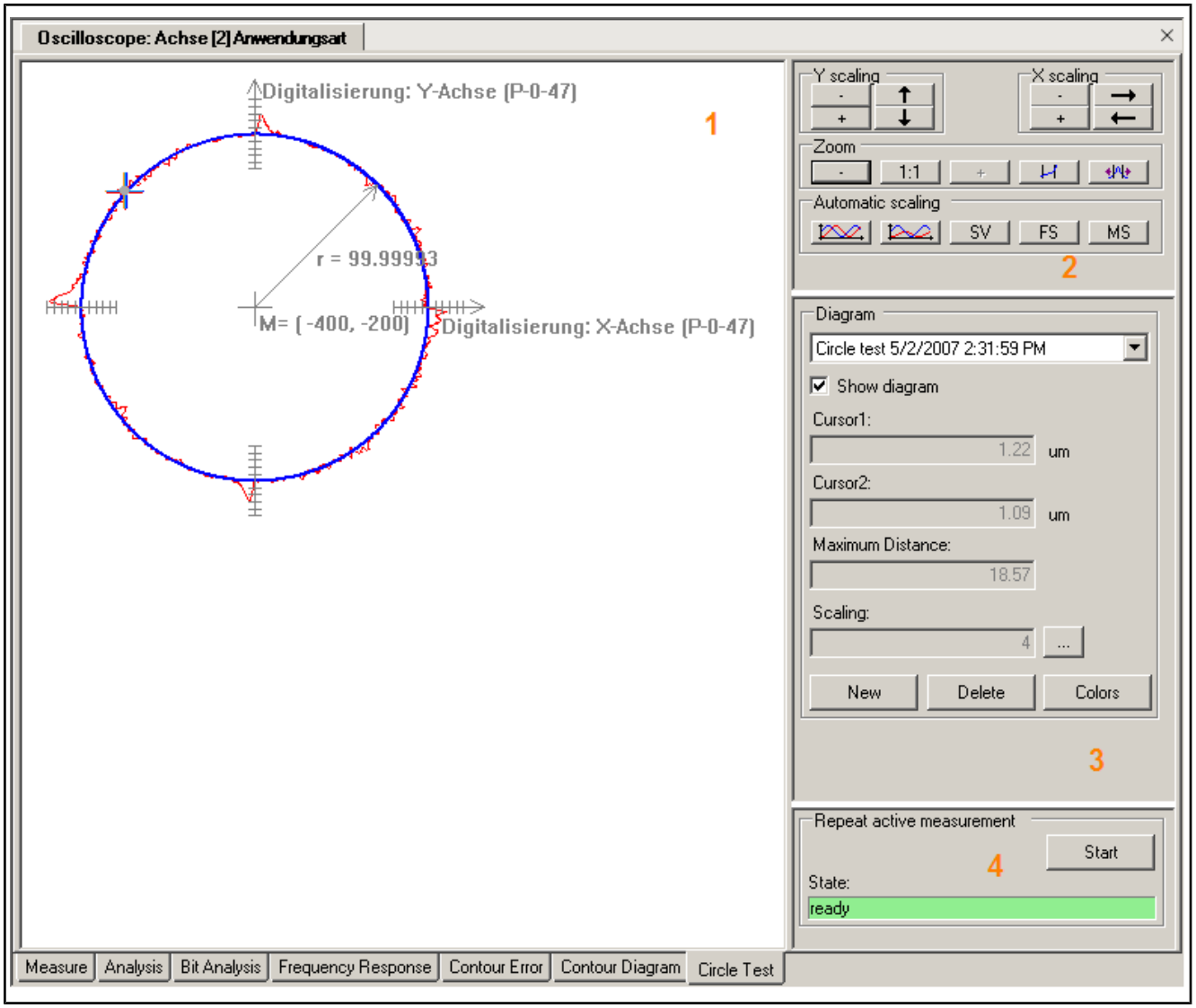


1 Graphical result
2 X or Y scaling and zoom
3 Operation with signal selection and cursor display
4 Repeat measurement and state display

Fig.3-11: Display of contour diagram

3.4.8 Circle Test

To view a circle test after a measurement select the following tab page:



1	Graphical result
2	X or Y scaling and zoom
3	Operation with signal selection and cursor display
4	Repeat measurement and state display

Fig.3-12: Display of circle test

3.5 Recording a New Measurement

3.5.1 General Information

You can perform a new measurement when you are connected to a device that supports the oscilloscope function.

Select the device and the signals, configure the measurement and the trigger, and start the measurement via the “Start” button. The measurement can be repeated.

3.5.2 Connecting Devices and Selecting Signals

Use the “Signals” button on the “Measure” tab page to open the “Configure Signals” dialog box.

Operation of the Oscilloscope Function

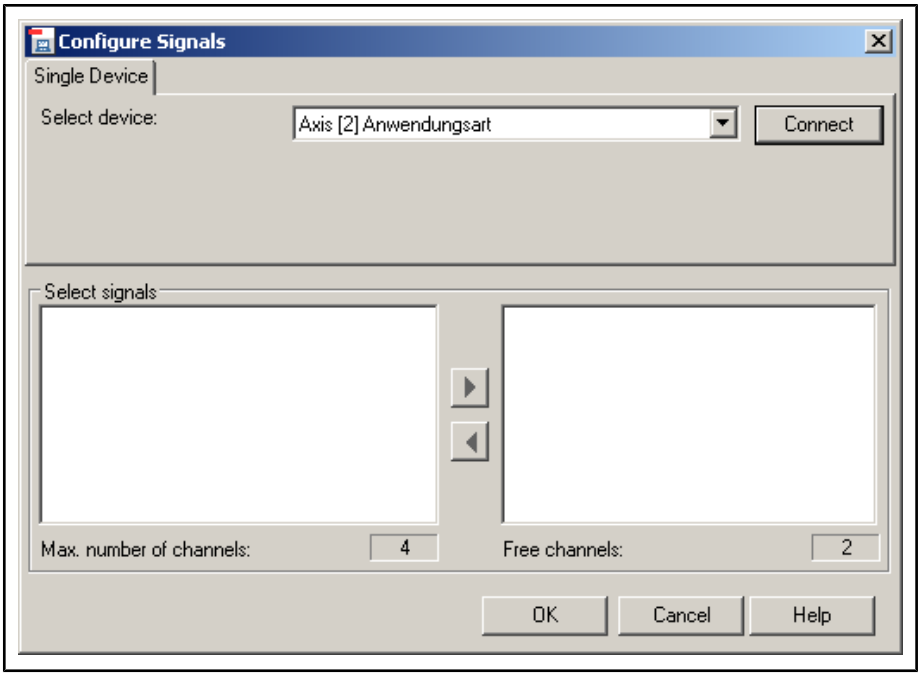


Fig.3-13: Configure Signals dialog box

Device Selection

Select the device to be connected and click on “Connect”.

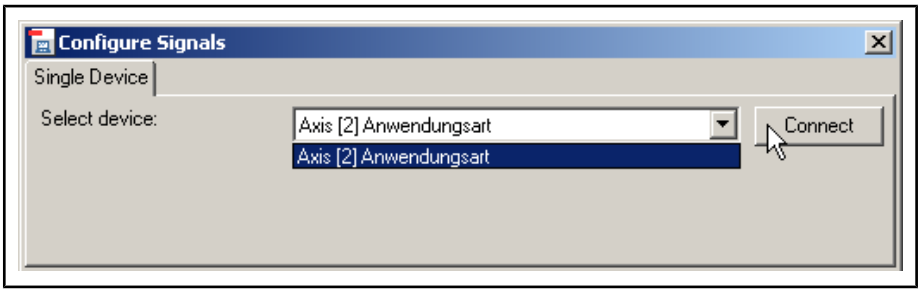


Fig.3-14: Configure Signals dialog box, select device

The text of the “Connect” button now changes into “Disconnect”.

Existing Signals

The list on the left contains all existing signals of the connected device. Select the signals whose measurement values you wish to record.

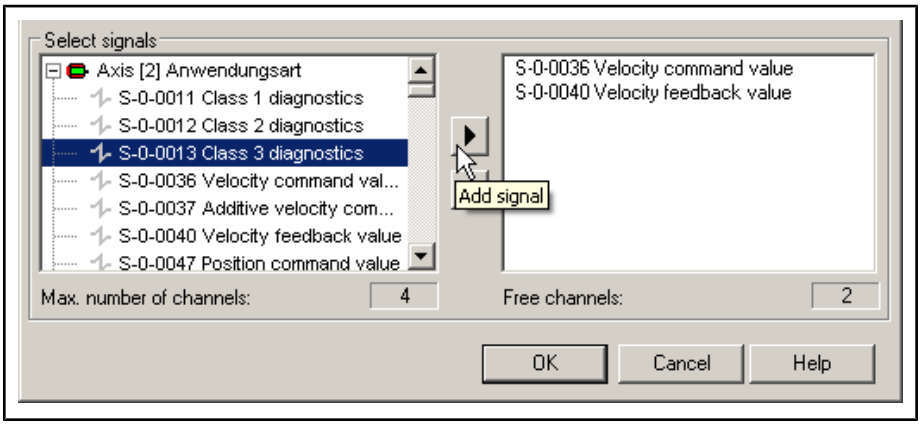


Fig.3-15: Configure Signals dialog box, select existing signals

Add

Use the “Arrow right” button or double-click to apply the selected signals to the list on the right (selected signals).

The number of signals allowed for measurement does not exceed the number of signals which can be processed by the connected device simultaneously (see fields “Max. number of channels” and “Free channels”).

Remove

To remove a signal from the right list (selected signals), select it and click on the “Arrow left” button or press <Del>.

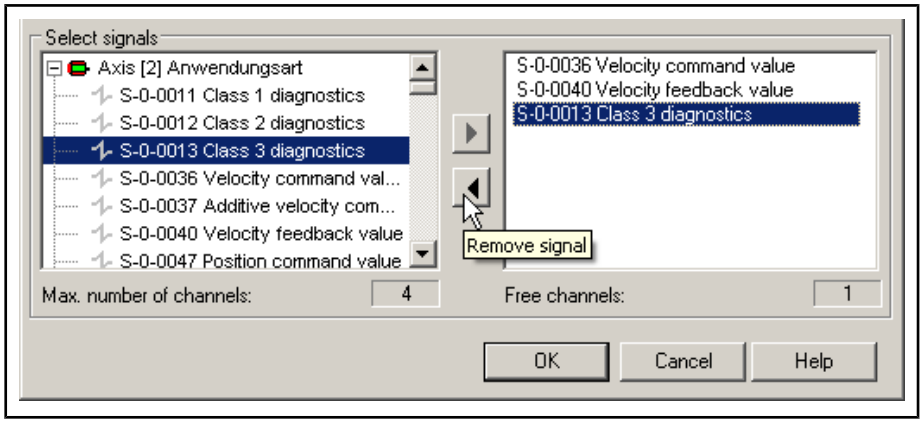


Fig.3-16: Configure Signals dialog box, remove selected signals

End of the Dialog

If you exit the dialog box with “OK”, the selected signals are applied to the signal overview.

As long as there are no measured data for these signals, the signals are inactive, i. e. they are displayed in gray font.

To discard your settings, click on “Cancel”.

The device selected remains connected after “Cancel”. To connect the oscilloscope to a different device, click on “Disconnect” and select the different device.

3.5.3 Configuring the Current Measurement

To change the memory depth and the time period of the measurement, click on the “Configure…” button in the “Configure measurement” area on the “Measure” tab page.

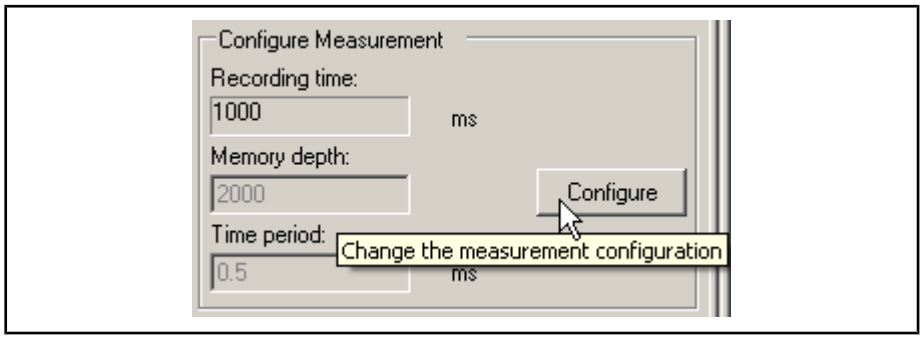


Fig.3-17: Configuring the Measurement

The “Configure Measurement” dialog box contains the default values for the device selected or displays the values saved last.

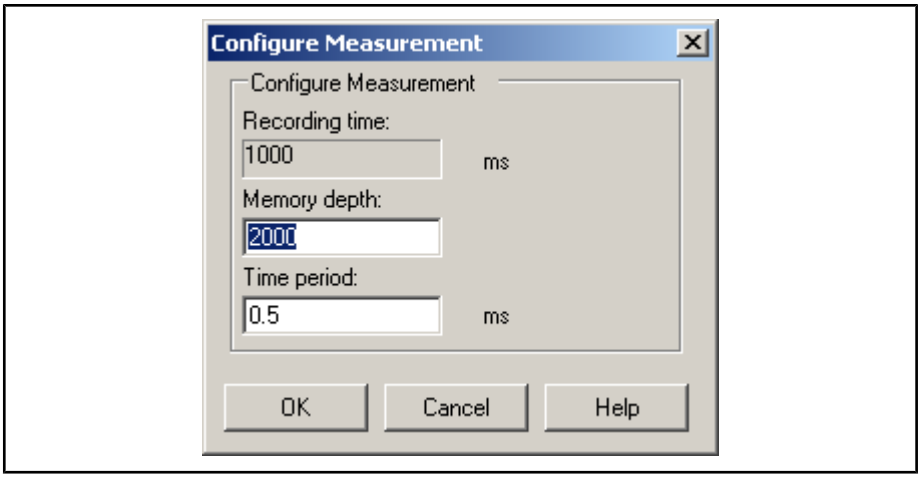


Fig.3-18: Configure Measurement dialog box

Recording Time This field displays the product from memory depth and time period. It is always recalculated after a change.

Memory Depth Enter the number of measurement points here.

The device provides the maximum memory depth. The lower limit is 1 % of the maximum memory depth. If your entry exceeds or falls below these limits, it will be corrected.

Time Period The device provides the possible time period values. If your entry differs from these values, it will be automatically corrected to the nearest value.

To save your changes, click on “OK”. Before saving, the limits will be checked and the values entered will be corrected, if necessary. The new measurement is completly configured now.

☞ If a measurement had been recorded or imported before, its signal data will be removed in the display.

To discard your changes, click on the “Cancel” button.

3.5.4 Configuring the Trigger

General Information

To customize the trigger settings, click on the “Configure” button in the “Trigger” area on the “Measure” tab page.

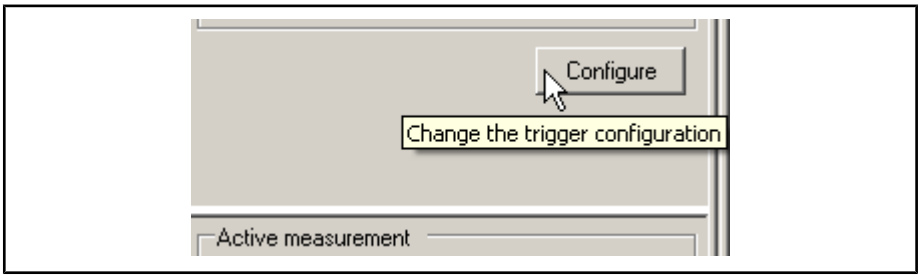


Fig.3-19: Configure Trigger button

This opens the “Configure Trigger” dialog box.

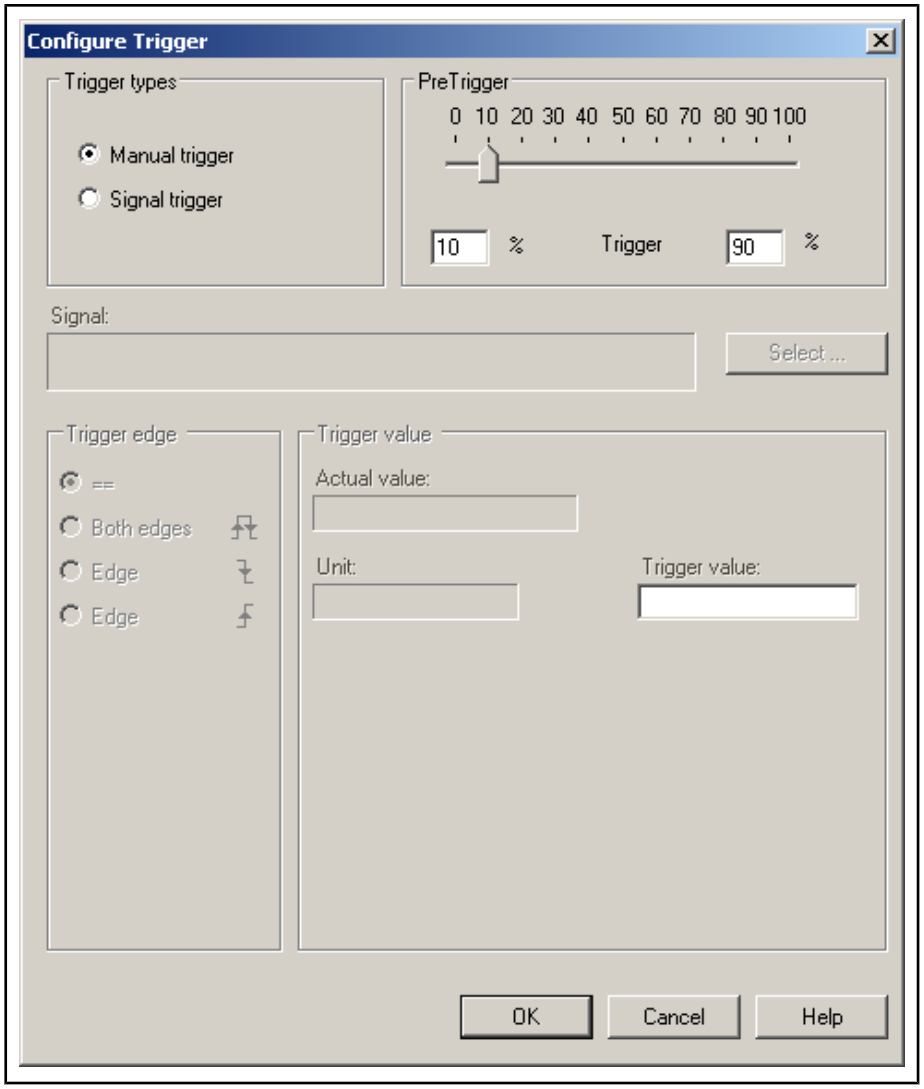


Fig.3-20: Configure Trigger dialog box

Manual Trigger

Selecting the Trigger Type

The trigger types supported by the connected device will be displayed in this dialog box.

In the following example the device connected supports the **manual trigger** and the **signal trigger**.

Further trigger types, supported by the current version, are:

- Automatic trigger
- Program trigger (only with MTX)

Select the “Manual Trigger” option in the “Trigger types” area and confirm via “OK”.

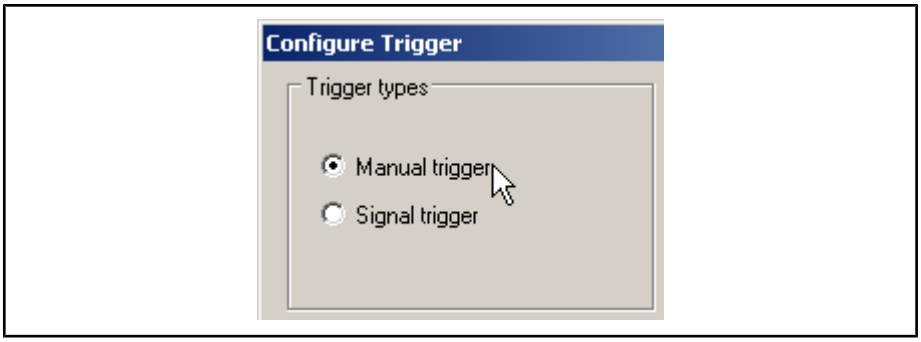


Fig.3-21: Remote Engineering Configuring the Trigger

Now the “Active Measurement” area will offer the “Trigger” button, as soon as the measurement has been started. Enable the trigger via this button. The measurement time is calculated from the memory depth and the time period. The data is displayed with the pretrigger taken into consideration.

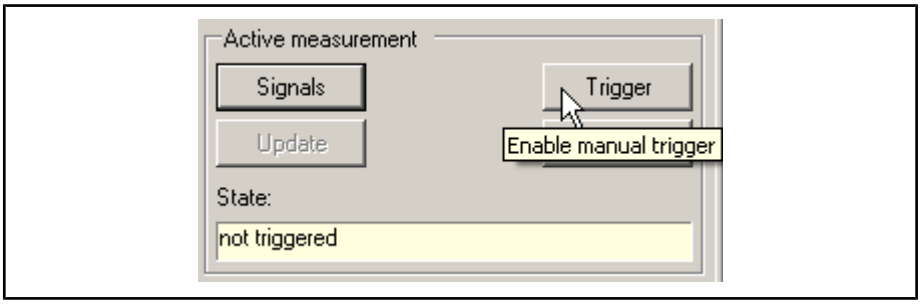


Fig.3-22: Trigger Manually button

☞ It depends on the device and its configuration, what trigger types will be offered. E. g., the manual trigger can be offered with IndraDrive, if the parameter P-0-0031 is configured non-cyclic.

Setting the Pretrigger

The pretrigger defines the percentage of the measurement to be called before the actual trigger signal. You can select a pretrigger value between 0 % and 100 %.

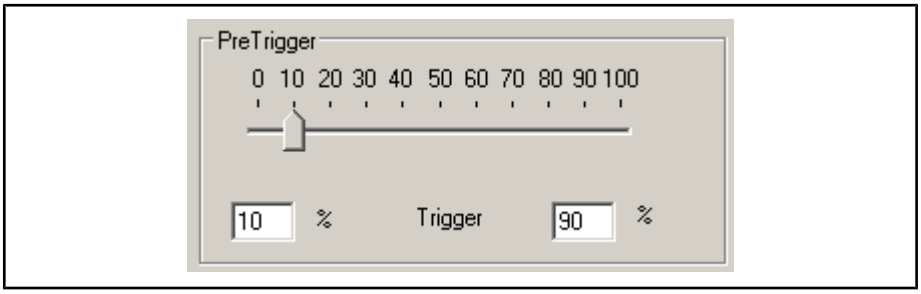


Fig.3-23: Select the pretrigger

☞ If the trigger is enabled before the time defined for the pretrigger is over, only a part of the desired pretrigger will be shown.

Example pretrigger

You have selected a pretrigger of 50%.

A memory depth (number of measurement points) of 4000 is defined.

According to these settings, 2000 values will be measured after the trigger has been enabled.

Result 1:

If the trigger is enabled after 2000 values have been measured at least, then 4000 values will be measured and displayed altogether (2000 values pretrigger + 2000 values after trigger).

Result 2:

If the trigger is already enabled after 1000 values have been measured, then only 3000 values will be measured and displayed altogether (1000 values pre-trigger + 2000 values after trigger).

Signal Trigger

Trigger Type “Signal Trigger”

If you intend to start recording a measurement depending on a signal, select the “Signal trigger” type. If the signal configured as trigger has exceeded or fallen below the set value, the trigger is enabled. The measurement time is calculated from the memory depth and the time period. The data is displayed with the pretrigger taken into consideration.

To select the trigger signal, click on the “Select” button in the “Configure Trigger” dialog box.

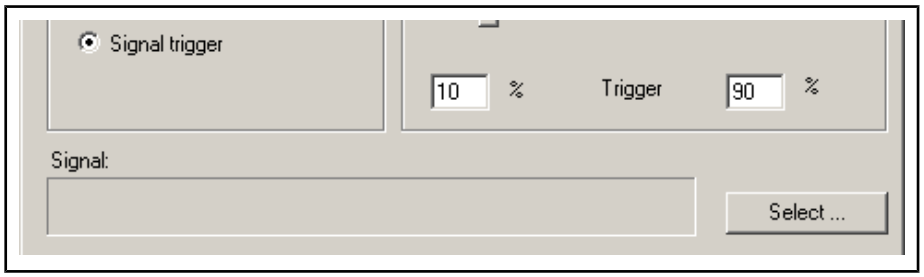


Fig.3-24: Configure Trigger dialog box, select signal trigger

Now all signals provided by the device for the signal trigger are displayed:

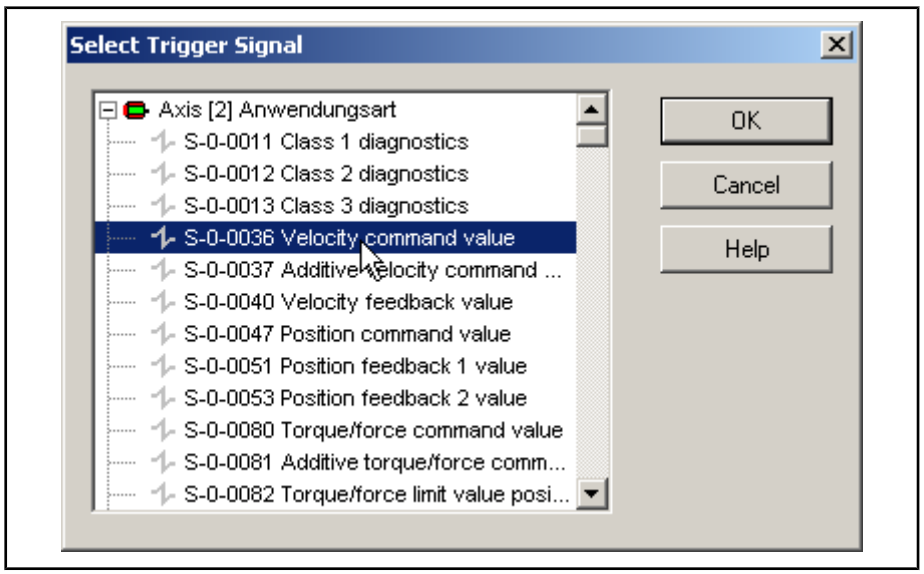


Fig.3-25: Select Trigger Signal dialog box

Select Signal

Select a signal from the list. To apply the signal, click on the “OK” button or double-click on the signal.

Cancel

If you do not wish to select a signal trigger, click on the “Cancel” button.

If you have selected a trigger signal, it will be displayed in the “Configure Trigger” dialog box.

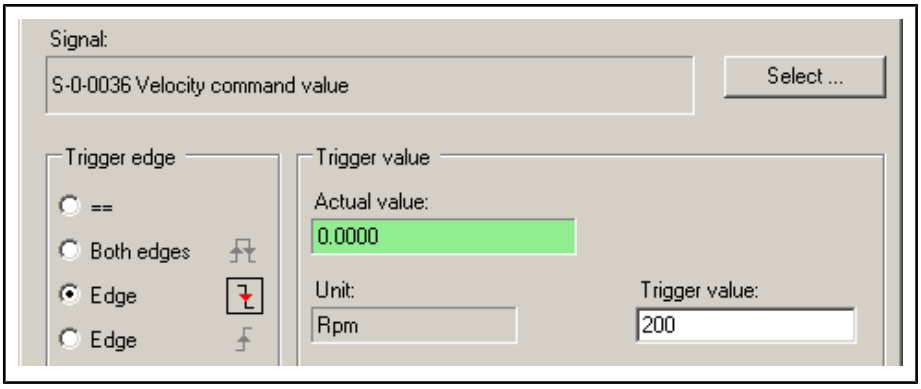


Fig.3-26: Configure Trigger dialog box, complete the settings

Trigger Edge This area provides only the trigger edges supported by the device. They show how the value to be entered is to be linked in order to enable the trigger.

Trigger Value Enter the trigger value to be reached or exceeded by the trigger signal in order to enable the trigger event.

☞ If the trigger threshold value is not reached during the measurement, you can adjust the trigger value even after the start of the measurement. You can use the actual value displayed in the trigger overview, in order to enter a suitable trigger threshold value.

Actual Value This box shows the current value of the trigger signal, thus you can enter a suitable trigger threshold value.

Unit This field displays the unit of the trigger signal.

Pretrigger Also with the trigger type “Signal Trigger” you can select a pretrigger (see chapter "Setting the Pretrigger " on page 158)

Triggering with Binary Signals

You can also select a binary signal as trigger signal. In this case, you can trigger to a specific bit and evaluate its state. Internally, the trigger mask is linked to the trigger value in a logic AND operation. If the result of the evaluation is a logic “1”, the trigger condition is met.

Example IndraDrive The “S-0-0013 Class 3 diagnostics” signal consists of 16 bits. For a complete meaning of all bits, please refer to the literature at the end of this description.

Here, bit 0 (velocity feedback value = velocity command value) and bit 1 (velocity feedback value < standstill window) are to be used for enabling the trigger.

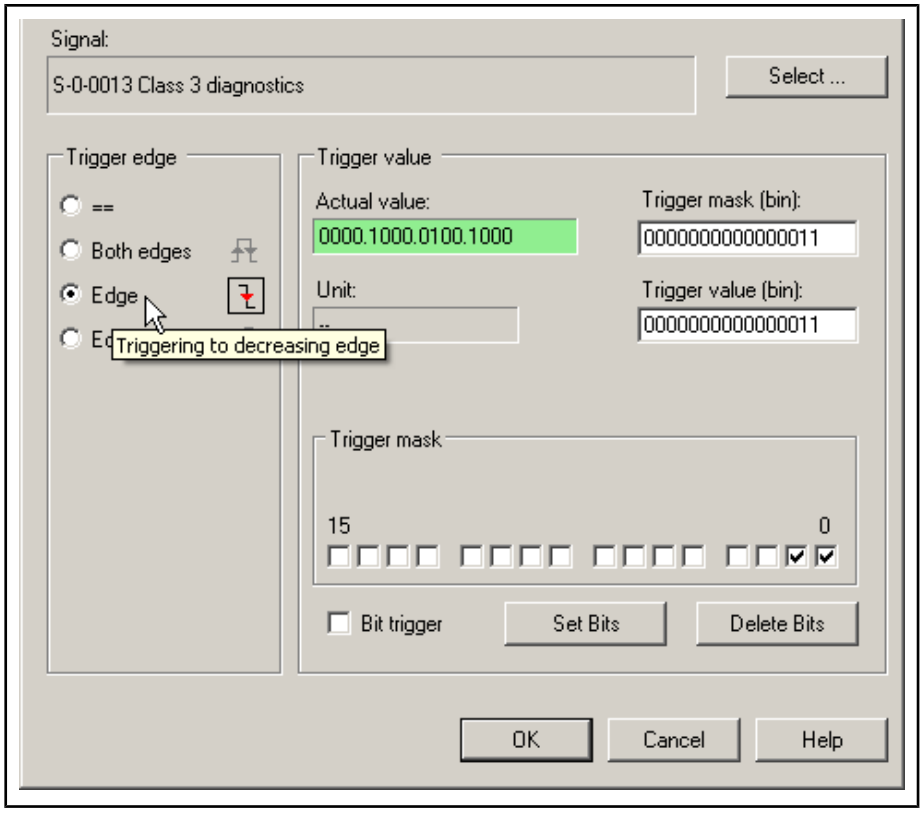


Fig.3-27: Configure Trigger dialog box, bit signal as signal trigger

You can define the mask bitwise by checking the individual check boxes or by directly entering the bits in the “Trigger mask (bin)” field.

Set Bits Click on this button to set all bits.

Delete Bits Click on this button to delete all bits.

☞ If you click on the individual bits, both the trigger mask and the trigger threshold value are adjusted for the logic AND operation. You can also change the trigger value manually, independently of the trigger mask.

☞ If several bits are selected, with a rising trigger edge, the trigger threshold value is 0 and cannot be changed. Thus any of the selected bits can start the trigger at a rising trigger edge.

Bit Trigger If you check this check box, the bit selection is limited to a single bit. In this case, the trigger mask and the trigger value are identical and will not be displayed. You can only trigger to falling or rising edges. The figure below shows the selection of a bit trigger.

Operation of the Oscilloscope Function

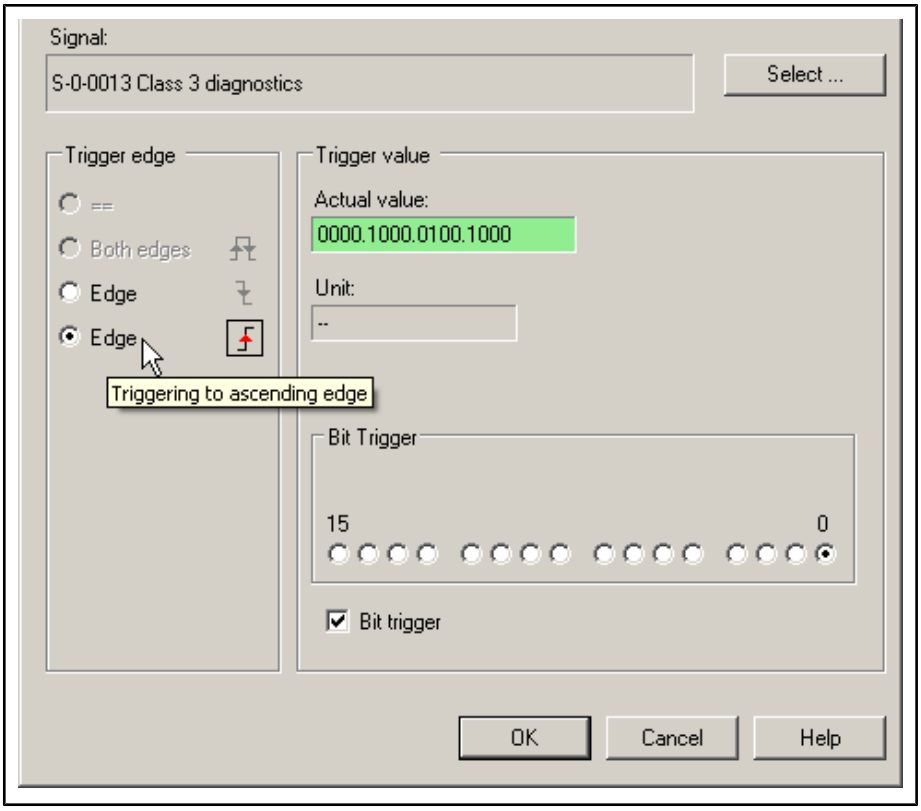


Fig.3-28: Configure Trigger dialog box, bit signal, bit trigger

The number of the selection points depends on the bit width of the selected trigger signal. It ranges from 8 bits to 32 bits.

3.6 Starting the Measurement

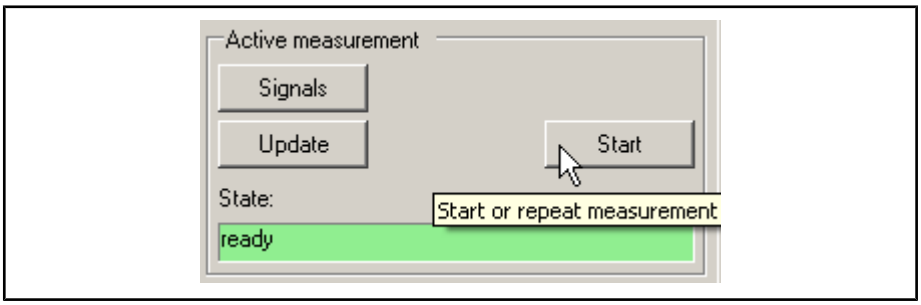


Fig.3-29: Start the measurement

Click on the “Start” button to start the measurement with the previously defined conditions. The field “State” shows the current state of the measurement.

If possible, the measurement lasts until the recording time defined in the “Configure Measurement” dialog box is reached.

The actual recording time can be shorter than the defined recording time. This occurs when the trigger value is reached before the time defined for the pretrigger ends.

After a successful measurement, the state is “ready” again and the measured values will be shown in the graphical display.

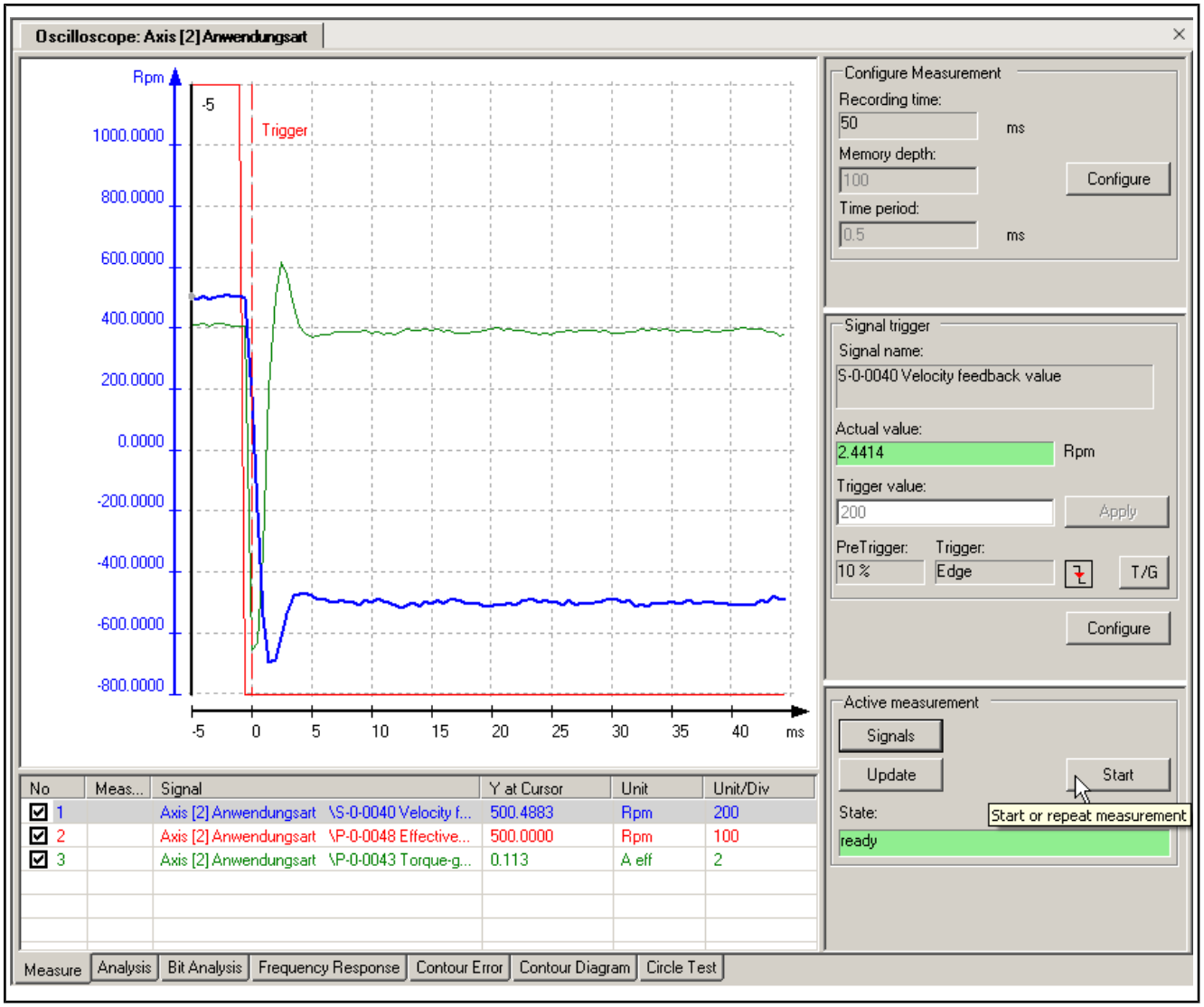


Fig.3-30: Oscilloscope, successfully completed measurement

If you start a measurement again, the last measured values will be replaced. In order to keep a measurement for documentation purposes, you have to save it via the **Oscilloscope** menu. See also chapter 3.10 "Loading and Saving a Measurement " on page 205.

3.7 Graphical Display

3.7.1 Signal Overview

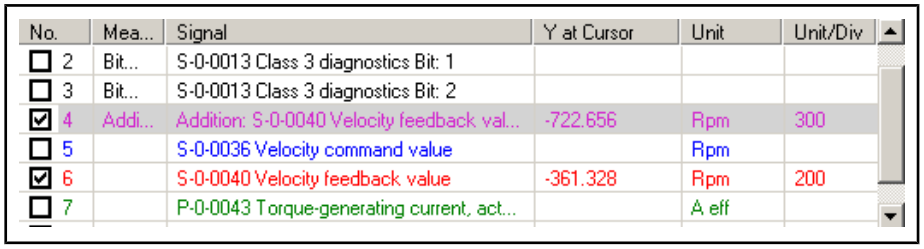


Fig.3-31: Signal overview

The check boxes allow you to show and hide signals.

The active signal has got a gray background. To activate a signal, click on it in the graph or in the signal overview.

☞ In the graph, an active signal is displayed with a width of two pixels; an inactive signal has a width of one pixel only.

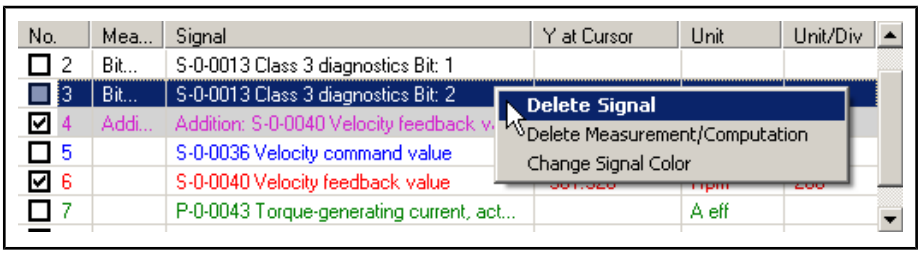


Fig.3-32: Context-sensitive menu of the signal overview

The context-sensitive menu offers further options to process signals.

Delete Signal

You can delete signals, which you have added before by **Diagnostics ▸ Load Signal Data** or **Add Signal Data** or computations.

☞ Currently measured signals cannot be deleted.

Delete Measurement / Computation

You can delete measurements, which you have added before by **Diagnostics ▸ Load Signal Data** or **Add Signal Data**. In this case, all signals belonging to the selected measurement will be deleted, measured signals as well as computed signals.

☞ You cannot delete a current measurement.

Change Signal Color

Use this menu entry to change the signal color of a signal. A dialog box for the color selection appears.

Selection of Several Signals

To select several signals in the signal overview, press the <Ctrl> key when clicking with the left mouse button. So several signals can be

- shown or hidden,
- deleted,
- get a different color.

3.7.2 Graphic

The graphic visualizes the measured values.

Cursor Display

The cursor has different appearances on a signal, on a line cursor or when zooming. See the following examples:

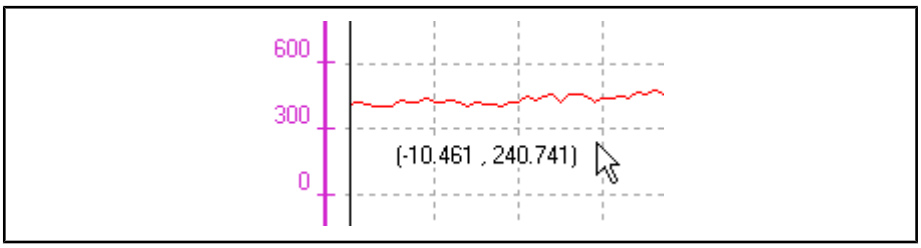


Fig.3-33: Default cursor

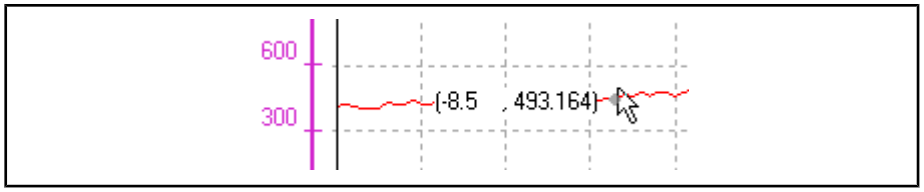


Fig.3-34: Cursor on a signal

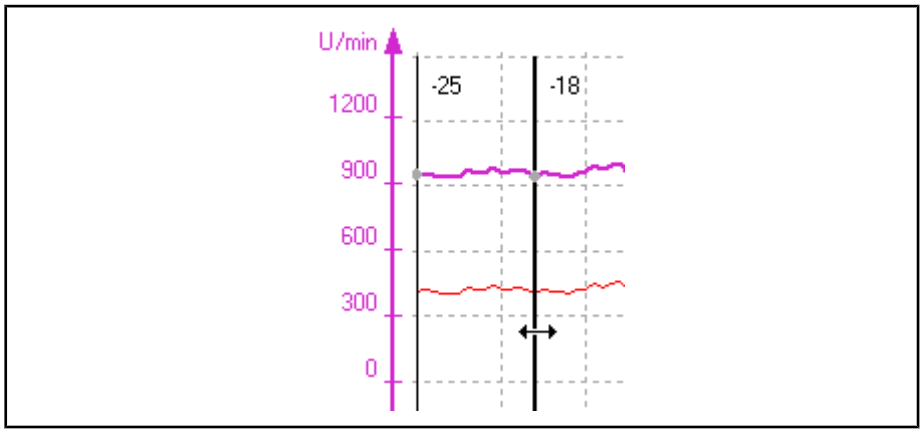


Fig.3-35: Cursor on a line cursor

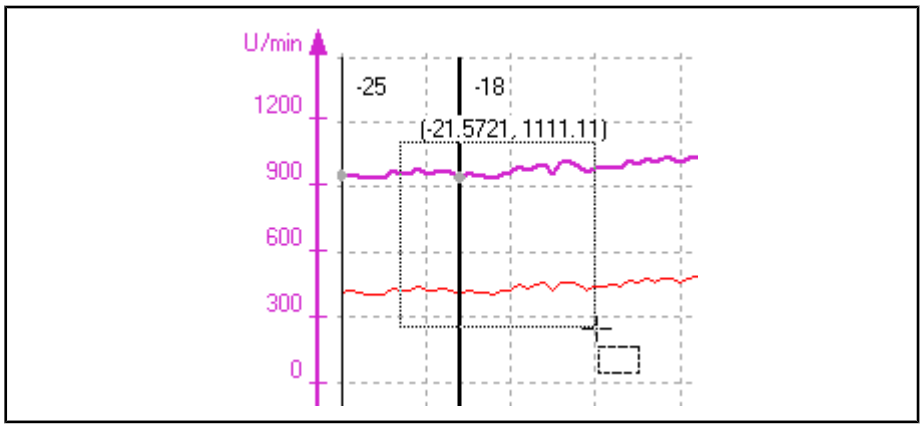


Fig.3-36: Cursor with zooming area

Trigger time

The interrupted red line in the graph indicates the time when the trigger has been enabled.

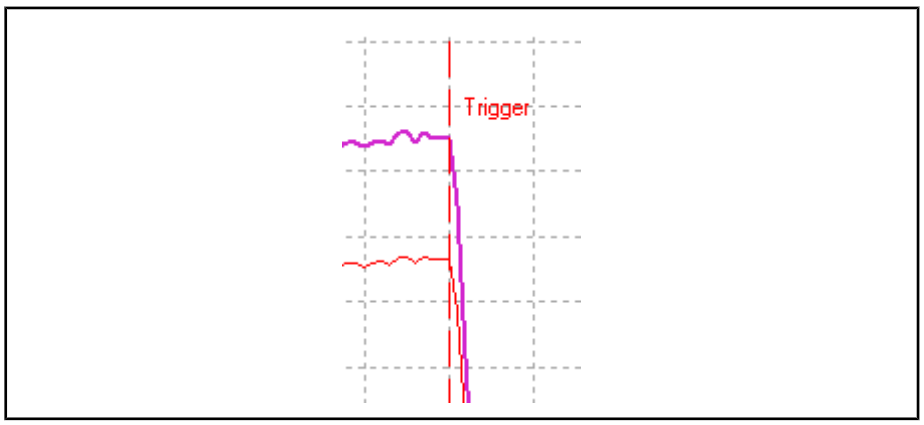


Fig.3-37: Trigger time

Line Cursor

The two continuous black vertical lines are the two line cursors. They can be moved horizontally with the mouse. The number next to the line cursor corre-

sponds to the value in the abscissa. The intersection point between the line cursor and the active signal is enlarged.

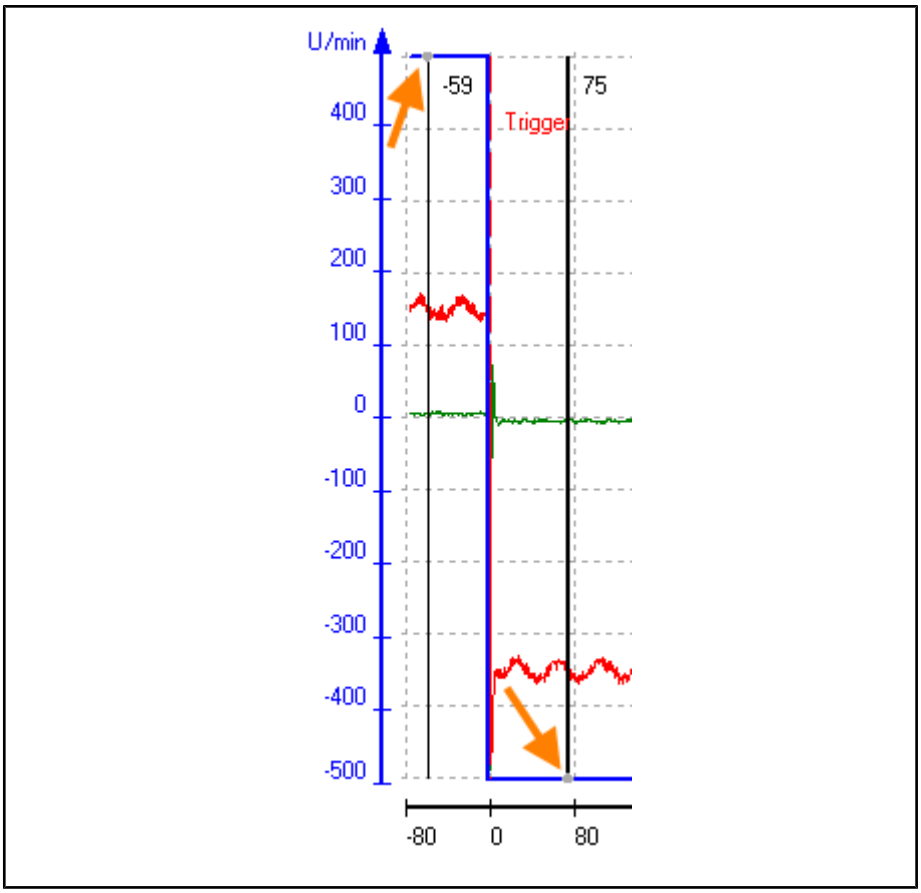


Fig.3-38: Display of the two line cursors

The line cursors can be moved with the mouse or with the arrow keys. The increment size depends on the defined time period.

☞ To move the active (wider) cursor in single increments, press the left or right arrow keys. Use <Ctrl> <Alt> together with the <Arrow Keys> to move it in steps of ten.

☞ Use <Ctrl> <Alt> <C> or click on a cursor in the graphic in order to select one of the two cursors.

3.7.3 Zoom

Zooming in an Area

To zoom in an area, left-click on the graph, hold the button and move the mouse. A rectangle will appear the size of which changes with the mouse movements. Once you release the mouse button, the zooming-in area is defined. It will be automatically sized to the entire graphical area.

☞ To start drawing the rectangle you have to select a free area on the graph. If the signals are so close to each other, that it is hard to find a free area, press <Ctrl> before clicking.

The figures below show the definition of an area and its result.

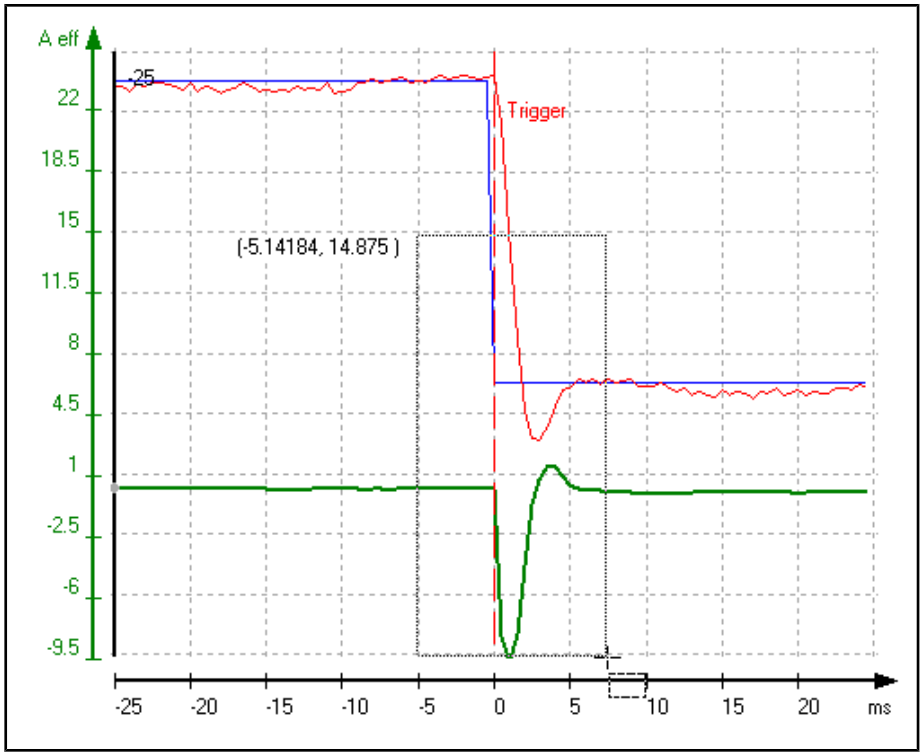


Fig.3-39: Display with zooming area

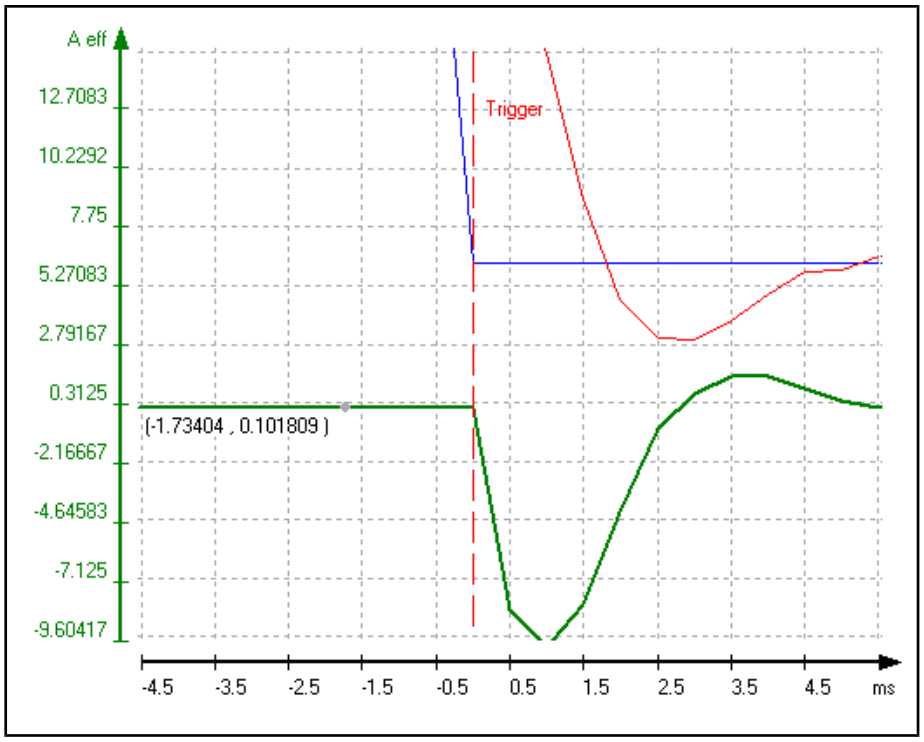


Fig.3-40: Display zoomed in

To further zoom in the area, repeat the steps above.

Zoom Levels

Each new definition of a zoom area is managed as a zoom level. To switch back and forth between these levels, click on the “+” and “–” buttons in the “Zoom” field.

Operation of the Oscilloscope Function

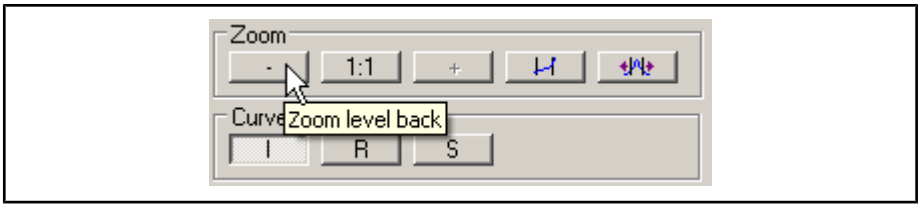


Fig.3-41: Zoom level down button

To return to the original graph, click on the “1:1” button.

Pressing the following shortcuts will also change the zoom levels:

- <Ctrl> <Alt> <P> corresponds to the “-” button
- <Ctrl> <Alt> <N> corresponds to the “+” button
- <Ctrl> <Alt> <Q> corresponds to the “1:1” button

Show Line Cursor

If one or both cursors are not in the zoomed area, they will not be positioned there automatically. To display the cursors in the current zoom area, use “Show Cursor”.

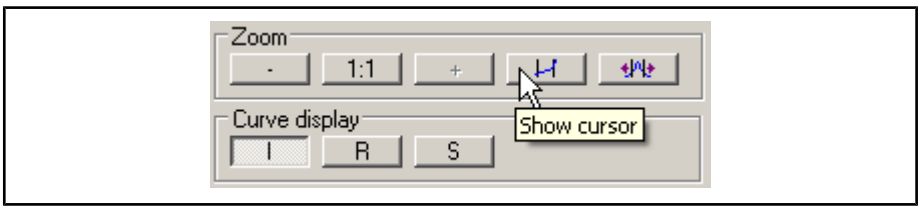


Fig.3-42: Show Cursor button

Zooming between the Line Cursors

Click on this button or press <Ctrl> <Alt> <Z> to enlarge the area between the two line cursors.

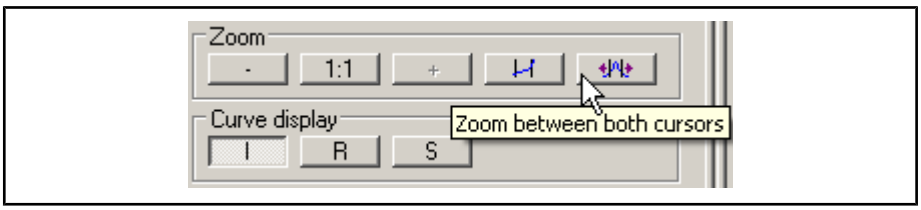


Fig.3-43: Zoom Between Both Cursors button

3.7.4 Moving Active Signal in Y Direction

You can move the active signal in Y direction by clicking on the “up arrow” and “down arrow” buttons in the “Y scaling” area.

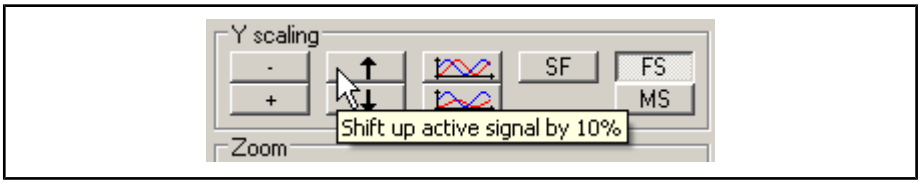


Fig.3-44: Move the active signal in Y direction button

3.7.5 Scaling Active Signal in Y Direction

Scale up the ordinate of the active signal in fixed steps by using the “+” button and scale it down by the “-” button in the area of the Y scaling .

3.7.6 Automatic support for scaling,

Absolute and Full Scaling

In absolute scaling, the signals shown depend on the largest signal.

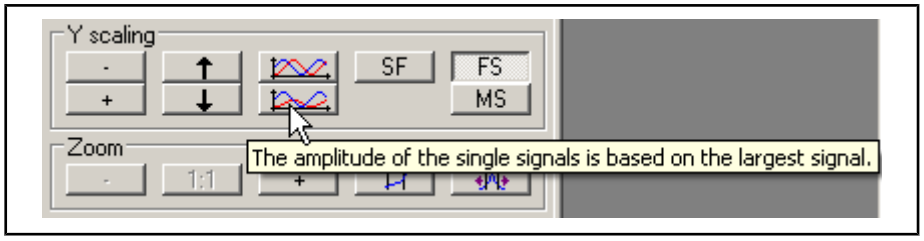


Fig.3-45: Absolute scaling

Use the button above the one for absolute scaling to scale up all signals to the full display area.

Example IndraDrive: Step Response with Current Profile

The figures below illustrate two different display types, with an example of three signals. The first figure shows the signals with absolute scaling.

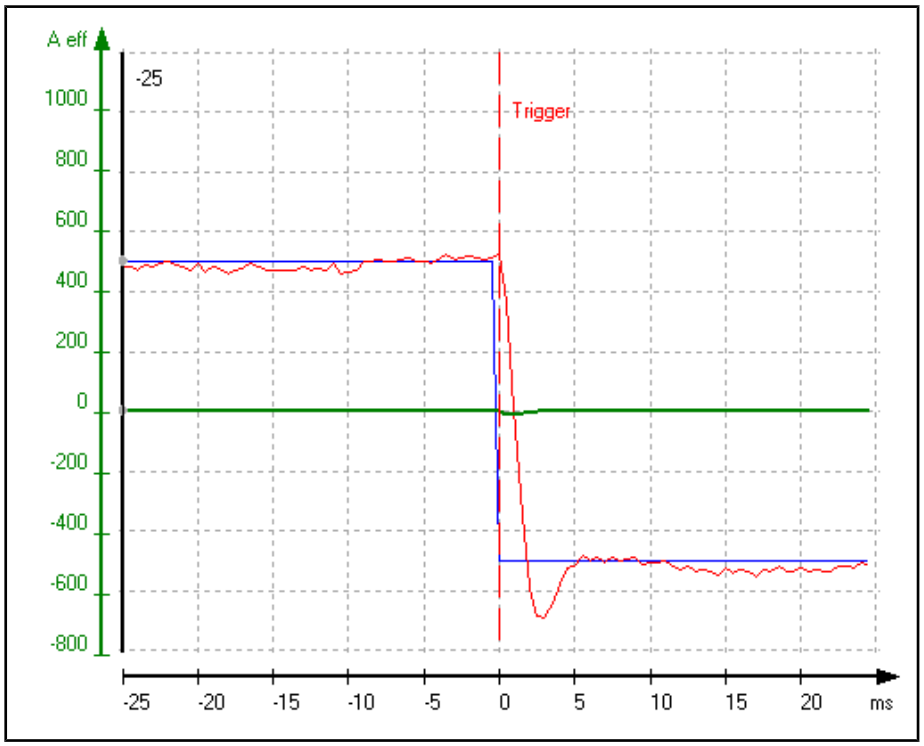


Fig.3-46: Display with absolute scaling

The next figure shows the signals with full scaling. All signals use the entire display area.

Operation of the Oscilloscope Function

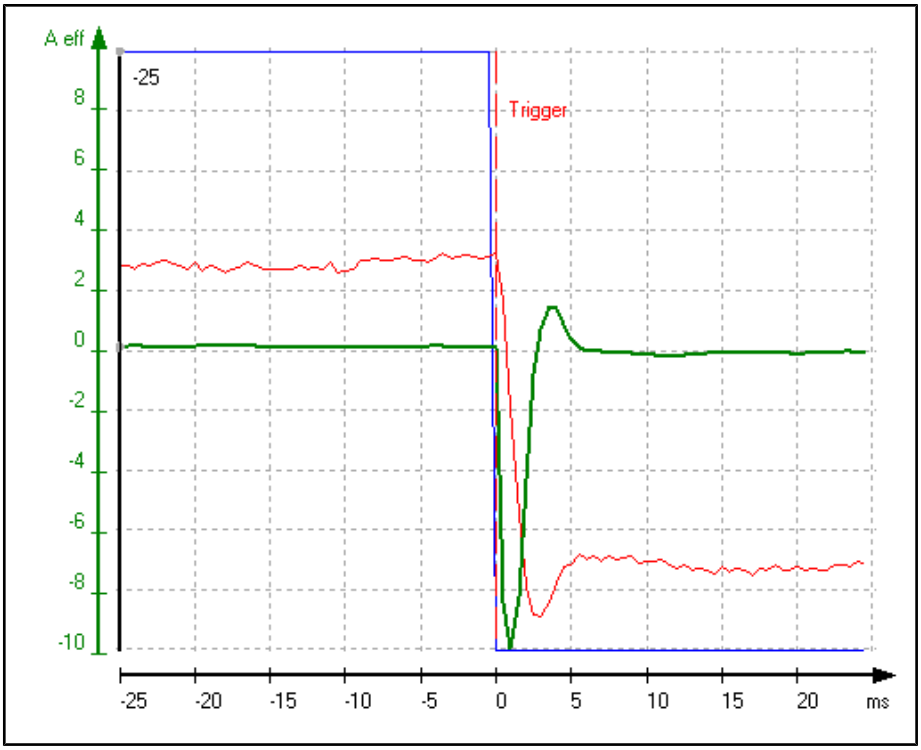


Fig.3-47: Display with full scaling

Enlarging the Active Signal in Y Direction

Use the “SF” button to enlarge the active signal to the entire display area.

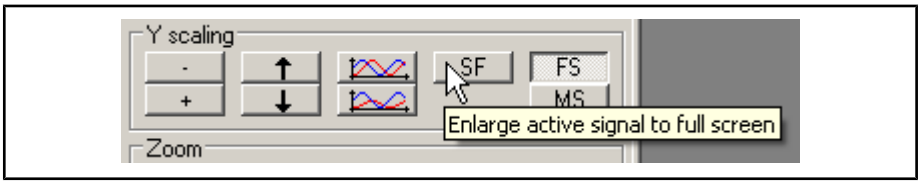


Fig.3-48: Enlarge the signal

Use this facility, if you want to enlarge a small signal to the entire display area in absolute scaling mode.

Manual Scaling

Beside the automatic scaling you can perform a manual scaling by “MS”.

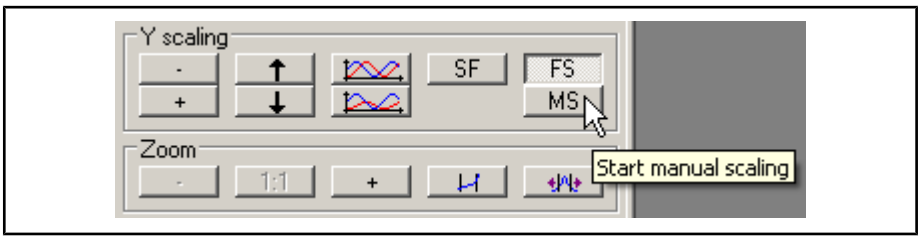


Fig.3-49: Start manual scaling

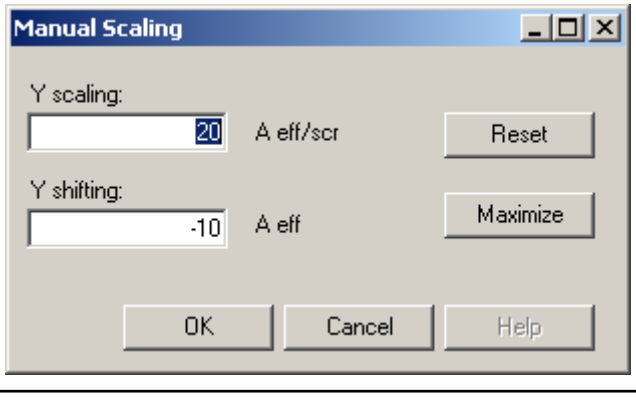


Fig.3-50: Manual Scaling dialog box

Having opened the dialog box, you can see the active values for Y scaling and Y shifting of the active signal.

To change these values click on the according input box and enter the desired values.

☞ The changes will be displayed in the graph immediately after leaving the input box. The dialog box will not be closed, thus you can correct errors, if necessary.

Close the dialog box and apply the changed scaling by “OK”.

“Cancel” closes the dialog box and discards the changes.

Use “Reset” to get the original values.

“Maximize” causes the full scaling of the active signal.

Fixed Scaling

The scaling of a measurement can be stored by “FS” for fixed scaling.

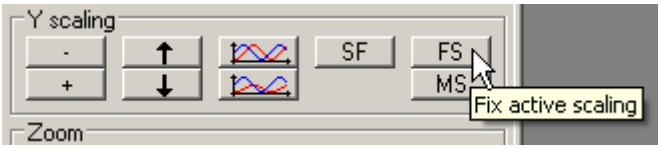


Fig.3-51: Fixed scaling

If you repeat a measurement, the scaling selected before is used.

☞ This also refers to a scaling obtained by zooming. That will optimize the process of performing several measurements consecutively.

3.7.7 Changing the Graphic Mode

Interpolated Measurement Curve

The single measured points are connected by lines, thus forming a measurement curve.

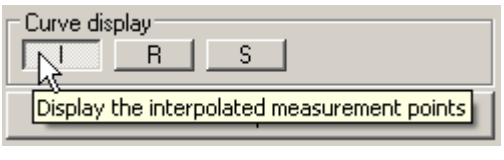


Fig.3-52: Display interpolated measurement curve button

Operation of the Oscilloscope Function

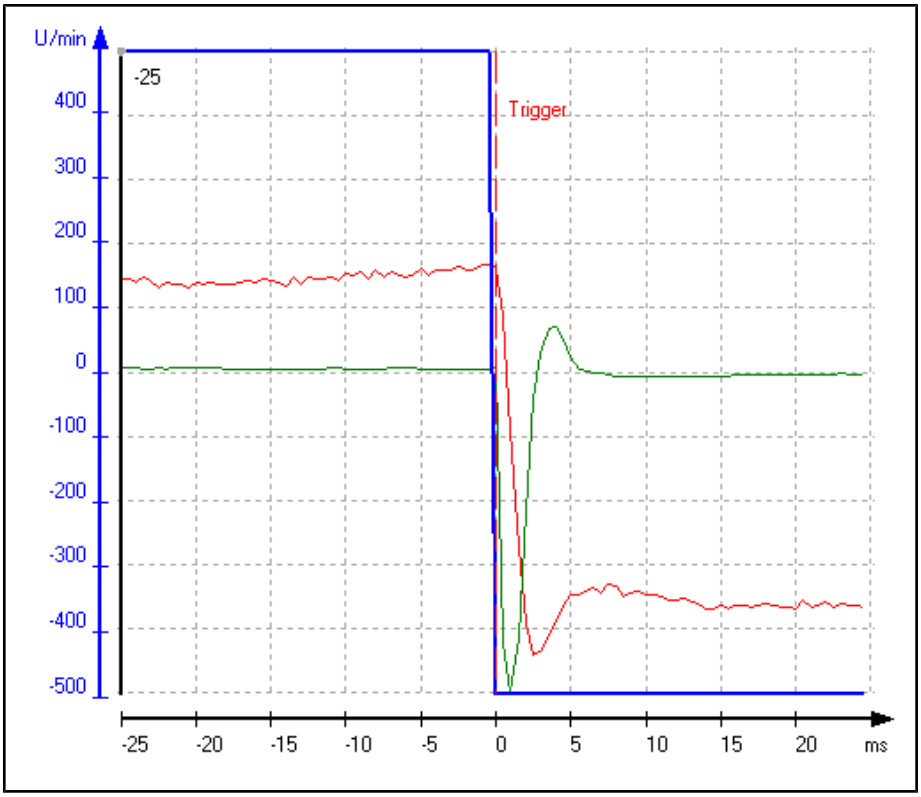


Fig.3-53: Connected measured points

Real Measurement Curve

The single measured points are connected by lines, thus forming a real measurement curve.

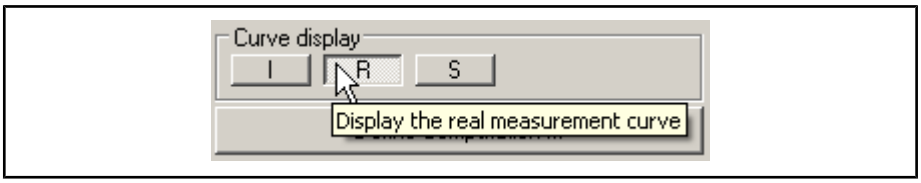


Fig.3-54: Display real measurement curve button

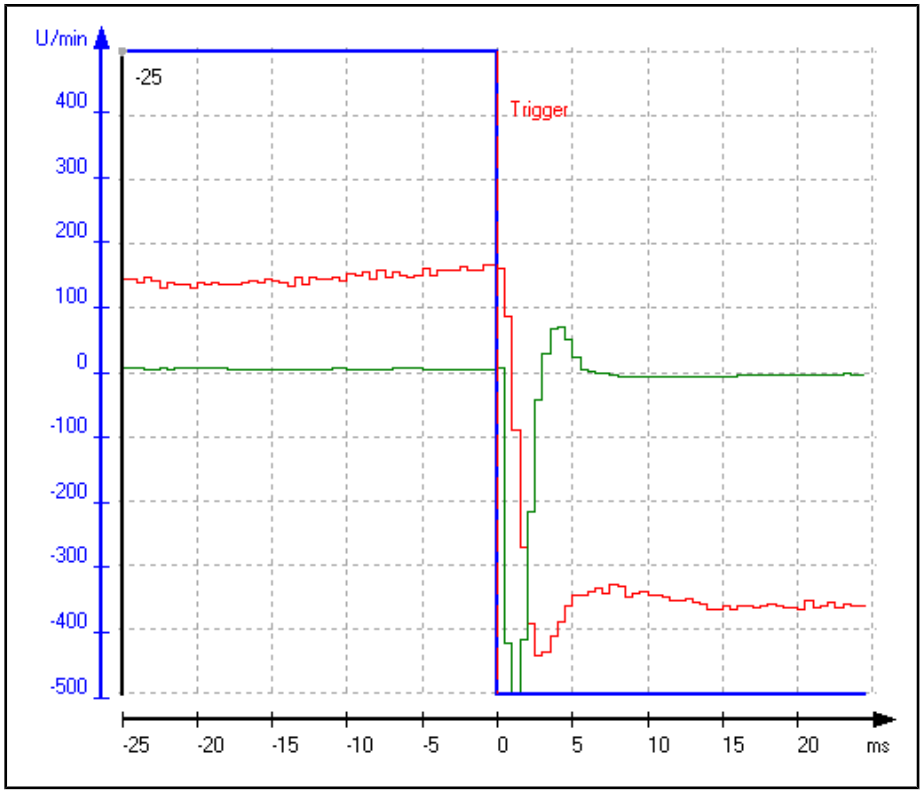


Fig.3-55: Real measurement curve

Display of the Single Measured Points

The single measured points are displayed and connected by lines, thus forming an interpolated measurement curve.

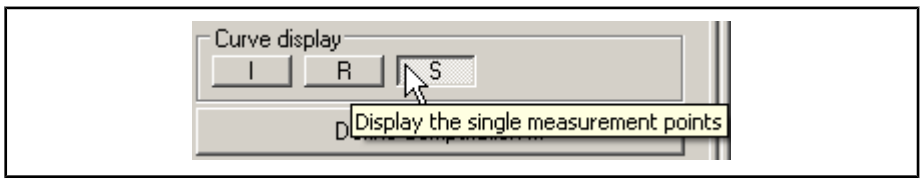


Fig.3-56: Display single measured points button

Operation of the Oscilloscope Function

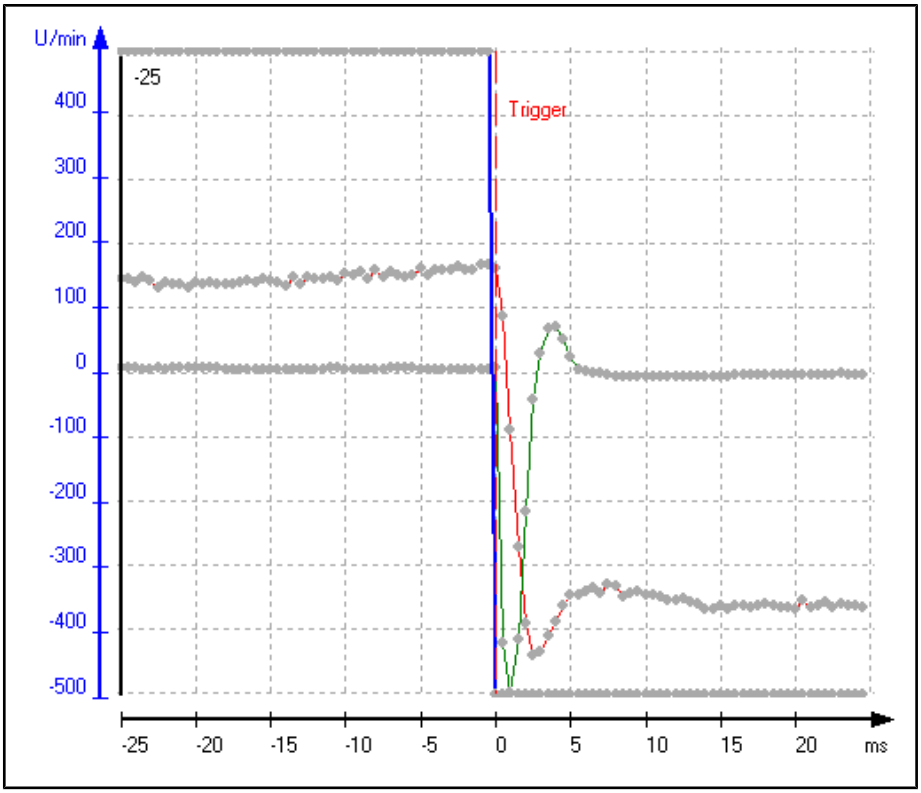


Fig.3-57: Display single measured points

3.8 Logic Analyzer

If signals with integer values (0x..., 0b...) are measured or added, then single bits can be extracted.

Example Select the signal “Class 3 diagnostics” additionally to the already selected signals, close the dialog box by “OK” and start the measurement.

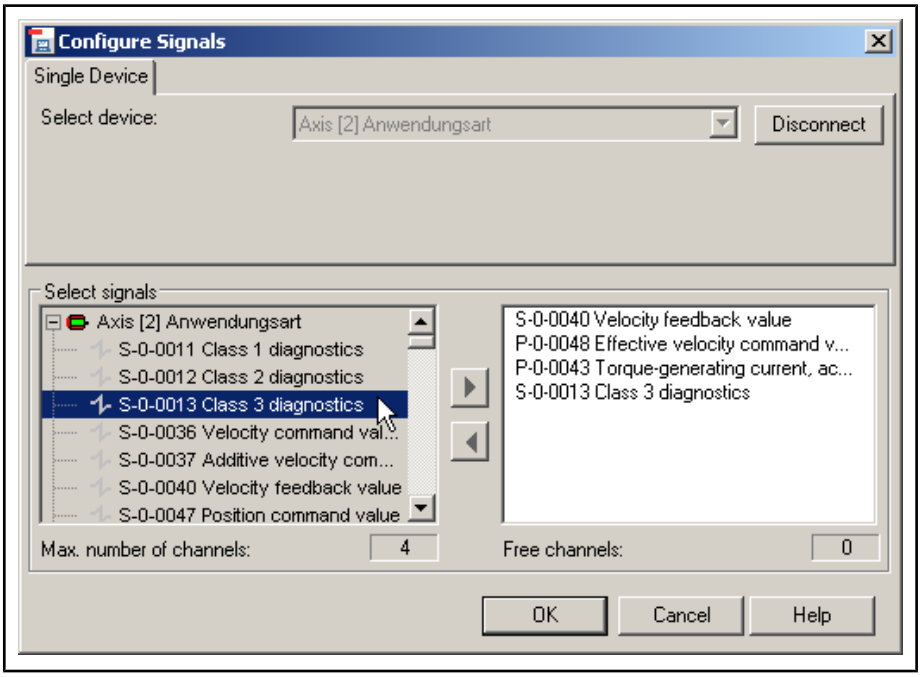


Fig.3-58: Configure Signals dialog box, selected bit signal

Go to the “Bit Analysis” tab and press “Select”. In the following dialog box select the “Class 3 diagnostics” signal and select the three lowest bits for display.

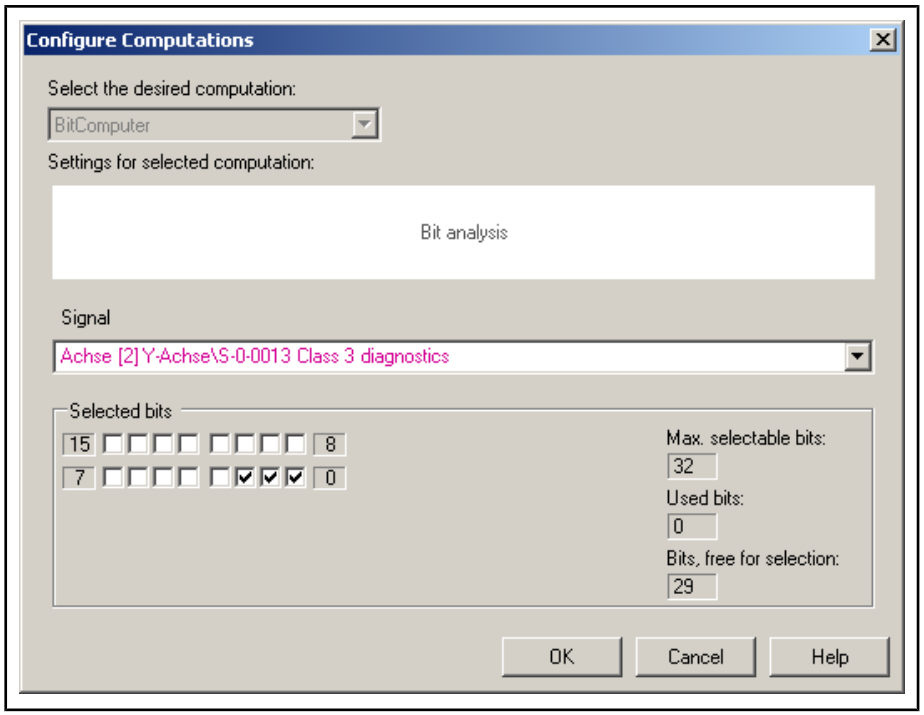


Fig.3-59: Configure Signals dialog box, selection of the bits

On each bit selection, the free and the used bits are counted. Close the dialog box by OK. Now the selected bits will be displayed.

The maximum number of selectable bits refers to all selected bit signals. You can select 32 bits.

Example IndraDrive

This example will show a measurement with bit signals.

Configuring the Trigger

For the example, select a signal trigger with the “S-0-0040 Velocity feedback value” signal, with falling edge, and with a trigger value of –200 rpm.

Configuring the Measurement

Select the memory depth of 100 measurement values within a time period of 0,5 ms.

The figure below shows the measurement result, with the bit signals not yet divided.

Operation of the Oscilloscope Function

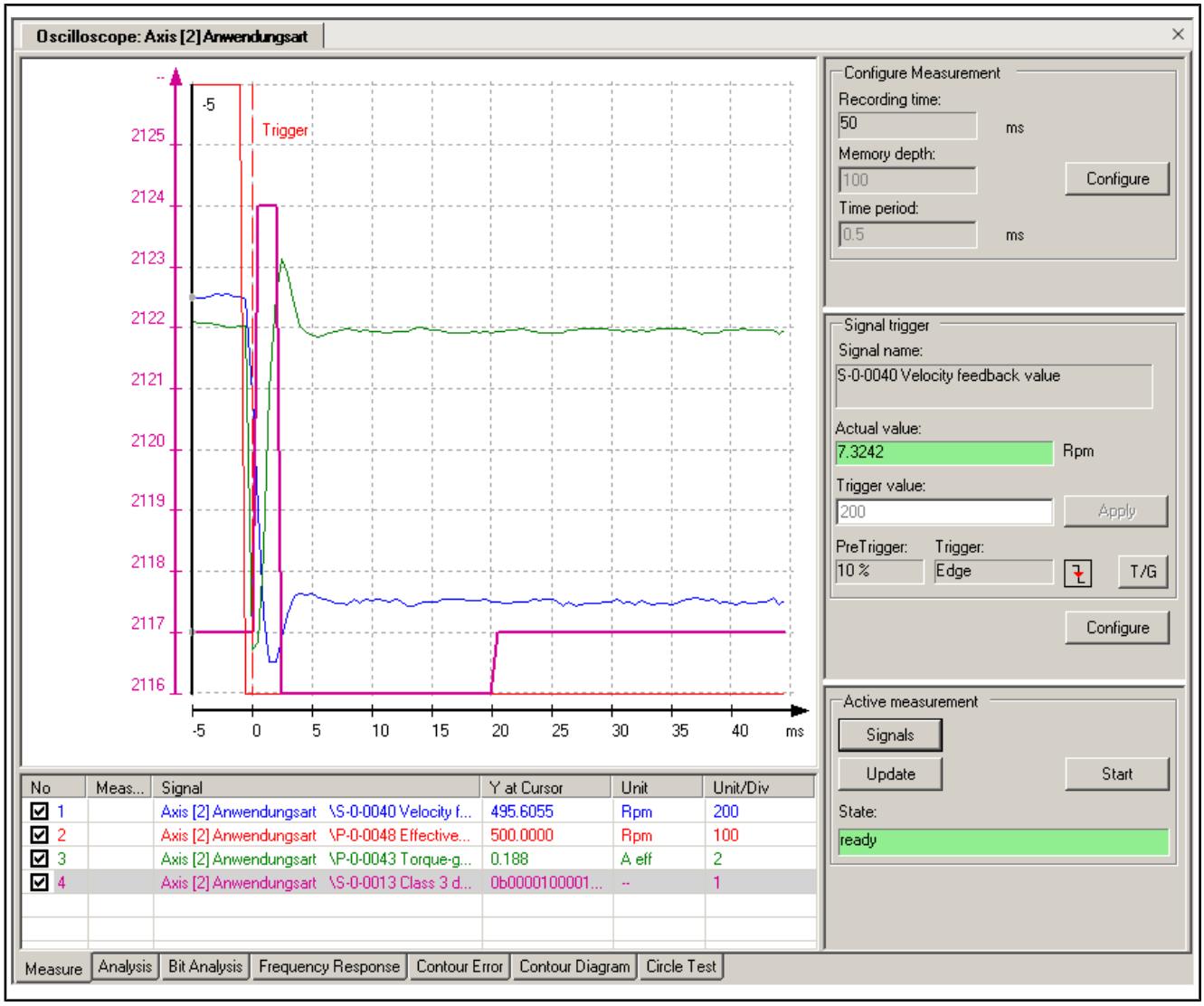


Fig.3-60: Display with comprised bit signal

Go to the “Bit Analysis” tab and press “Select”. In the following dialog box select the “Class 3 diagnostics” signal and select the three lowest bits for display.

The figure below shows the result of the bit computation.

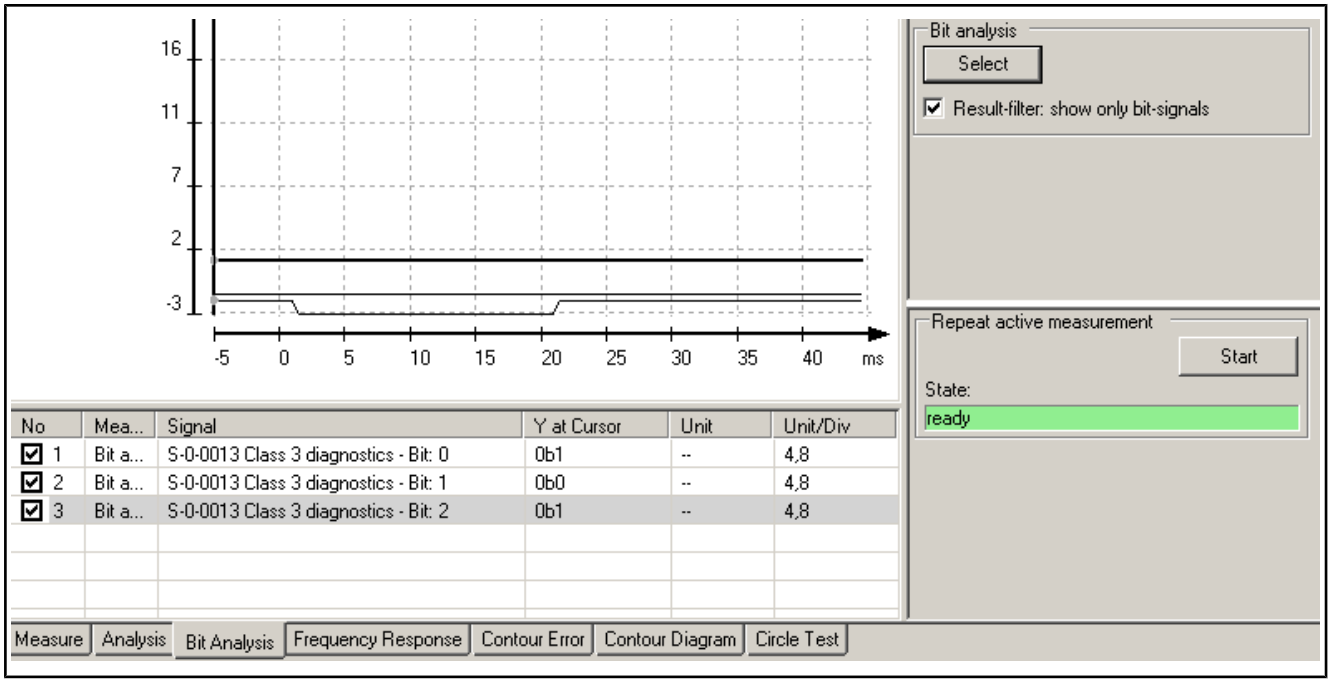


Fig.3-61: Display of single bit signals

Only the computed bit signals are displayed. The other signals have been hidden by setting the check mark and thus activating the filter function.

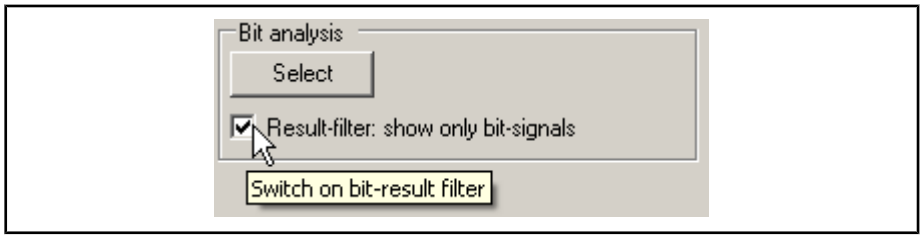


Fig.3-62: Activated filter function for bit signals

Graphical Interpolation

The rising edge of the lowest signal is not a vertical line. This is caused by the fact that the values are linearly interpolated and represented as a curve. This is reasonable for “analog” signals, but not for bit signals.

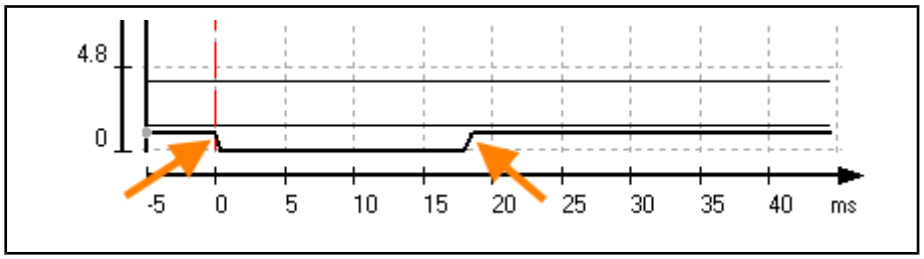


Fig.3-63: Display of bit signals with inclined edge

To obtain a digital graph, go to the “Analysis” tab page. Select the display of the real measurement curve by “R” in the “Curve display” area.

Operation of the Oscilloscope Function

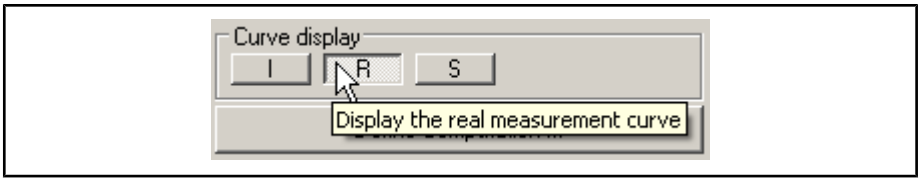


Fig.3-64: Display of real measurement points

The figure below shows the result:

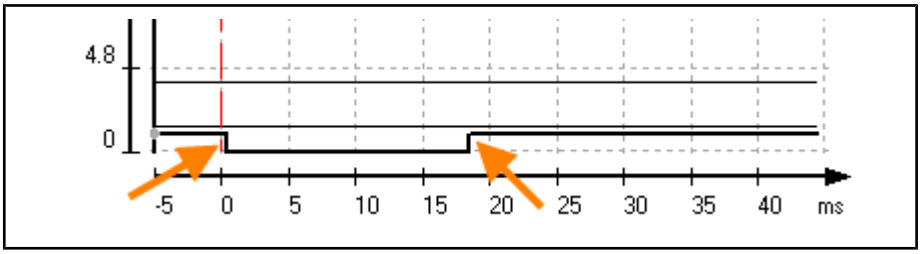


Fig.3-65: Display of real measurement points with rectangular edge

Display of all signals

Use the checkbox to disable the filter function of the display of bit signals.

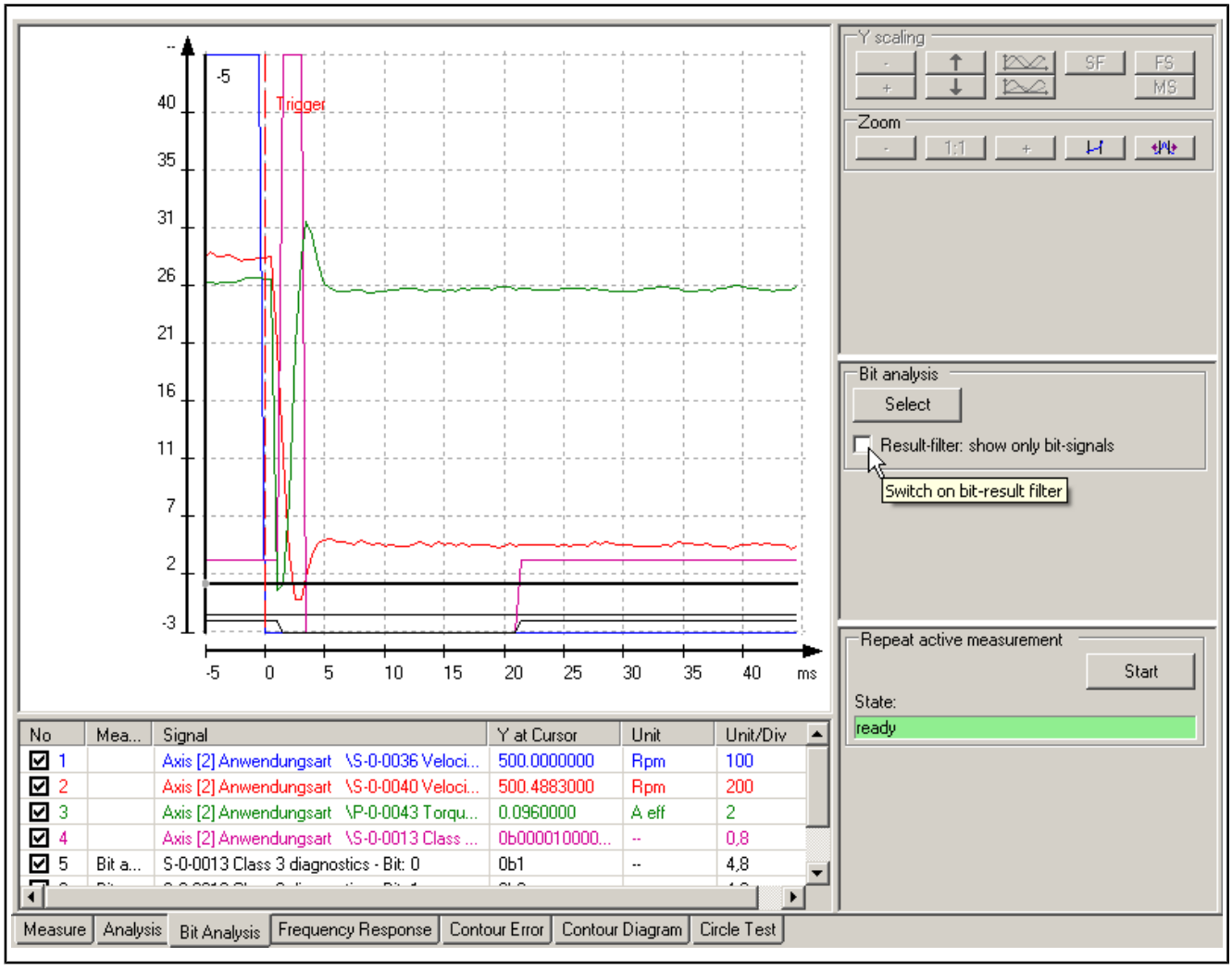


Fig.3-66: Display of all signals

3.9 Computations

3.9.1 General Information

Measured signals can be combined with each other mathematically, in order to obtain further results. Select the desired computation and then the signals to be combined. The signal overview and the graph will display the signals computed.

☞ If a measurement is started again, the computations will be repeated automatically.

☞ Having configured a computation and started the measurement, you will obtain the signal data as well as the computation data. These data can be saved together. The configuration of the computation will be saved, too.

☞ To change an already configured computation, press the <Ctrl> key when clicking on the “Define Computation ...” button on the “Analysis” tab page.

The examples below will show possible computations with the oscilloscope.

3.9.2 General Computation Formula

Example with Position Feedback Value S-0-0051

This example with IndraDrive shows, how to compute the velocity using the position feedback value.

The following settings have been specified:

- 4000 measurement points
- Time grid of 0.5 ms
- Signals S-0-0036, S-0-0040 and P-0-0043
- Further signal S-0-0051 position feedback 1 value
- Signal trigger with S-0-0040 velocity feedback value

Furthermore, the command value has been activated by a periodical square-wave signal of the drive internal command value generator:

Operation of the Oscilloscope Function

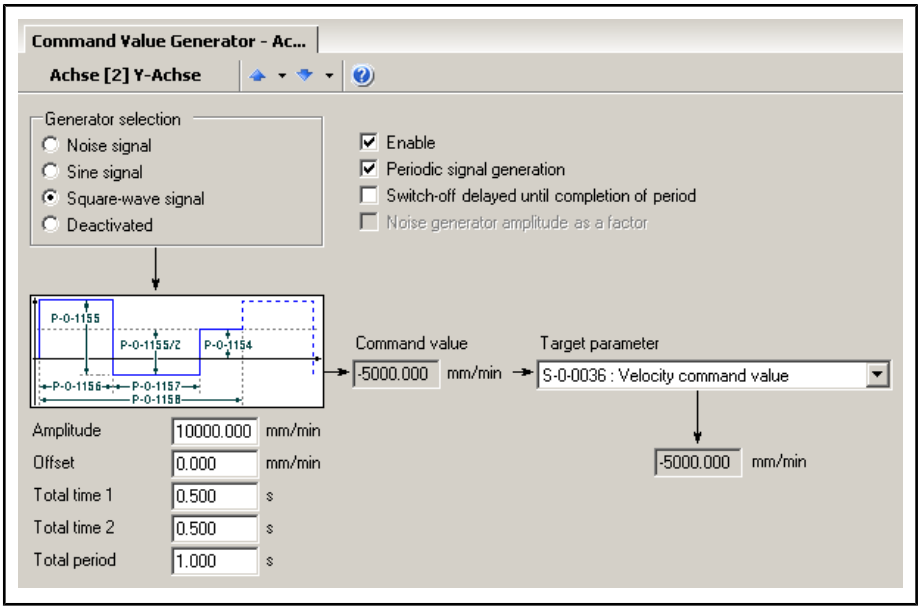


Fig.3-67: Defined command value

☞ You can change the settings only in the de-activated state. They will be sent to the drive immediately after the generator, e. g. the square-wave generator, has been activated.

After the measurement you will see the following curves:

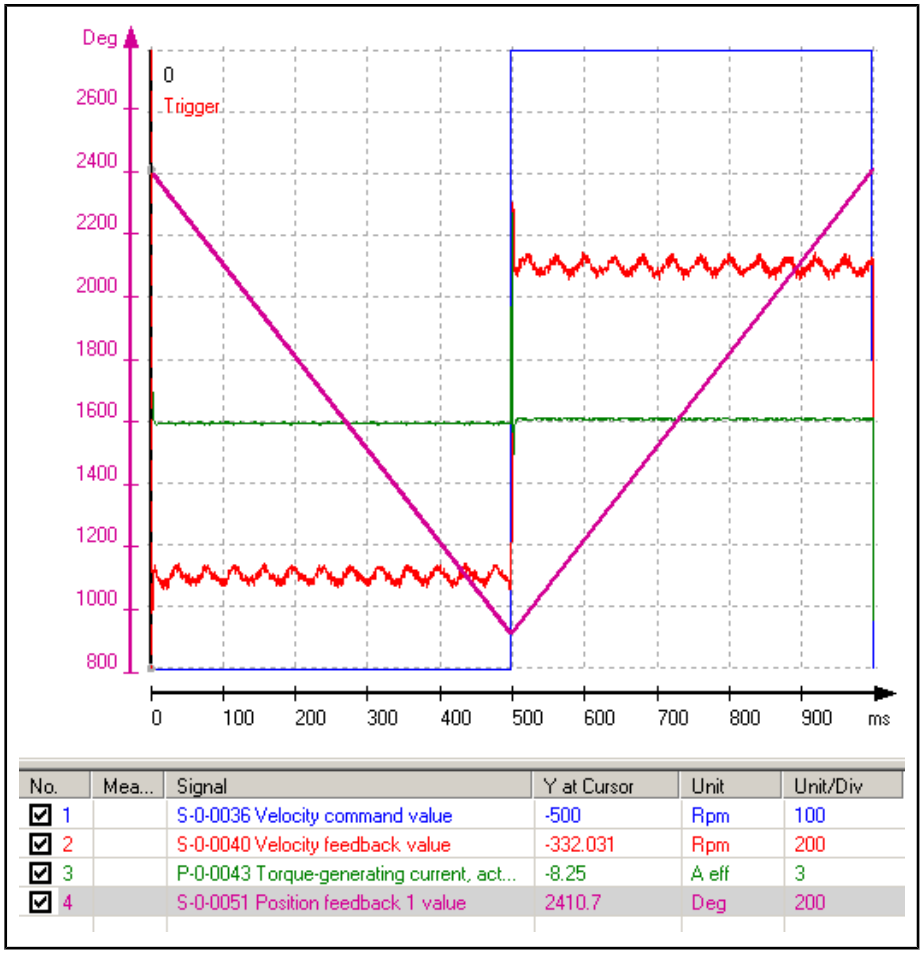


Fig.3-68: Result of the measurement

Computing

Start the computation in the “Analysis” tab page.

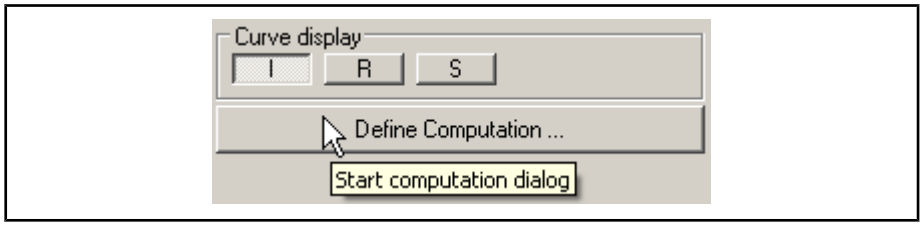


Fig.3-69: Define Computation button

The “Configure Computations” dialog box will be opened.

Operation of the Oscilloscope Function

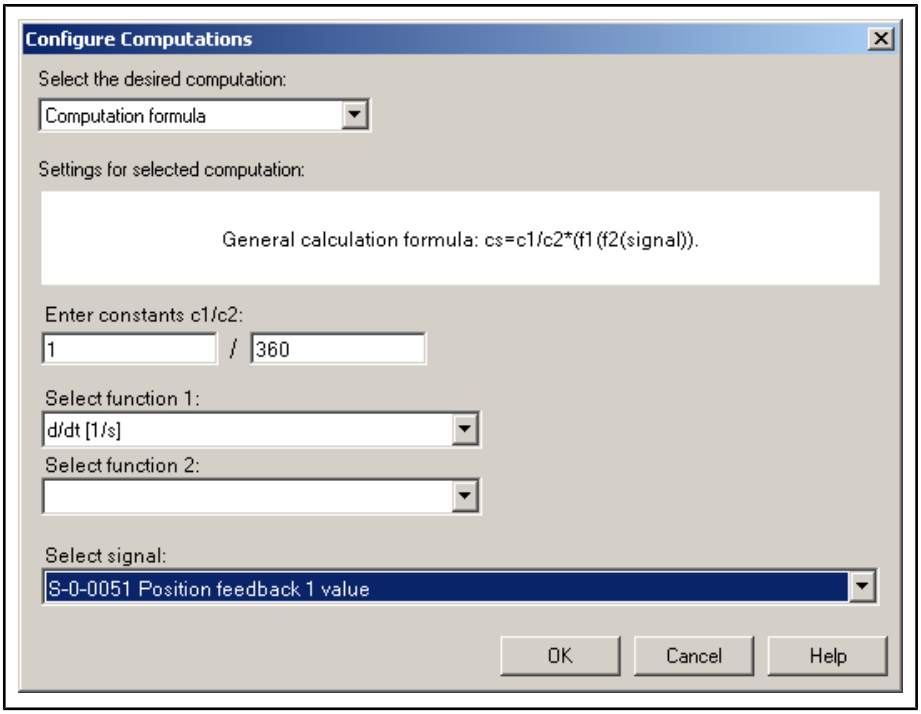


Fig.3-70: Configure Computation dialog box

1. The parameter S-0-0051 position feedback 1 value is given in degrees. To obtain the number of rounds define the constant c2 = 360, and thus divide the result by 360.
2. Select the derivation d/dt (1/s) as function 1.
3. Select signal S-0-0051.
4. Start the computation by “OK”.

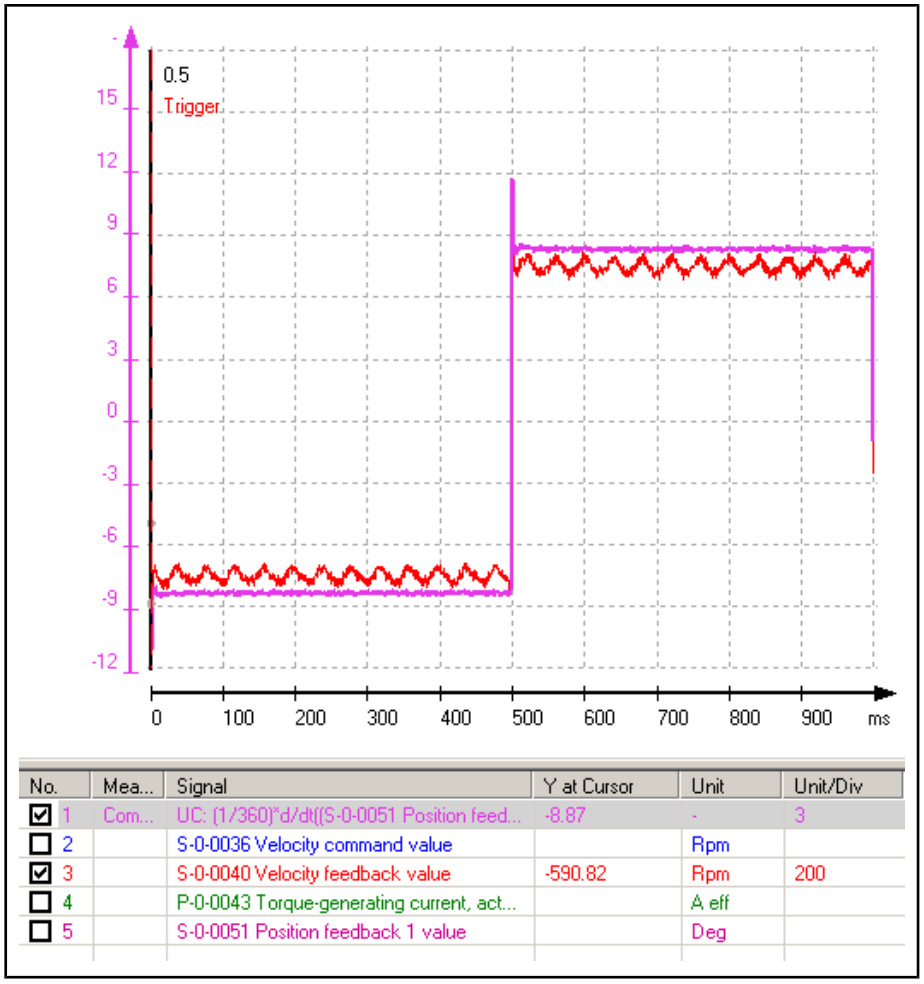


Fig.3-71: Result after derivation d/dt (1/s)

The result of this computation, 8.87 rounds per second, is shown in the signal overview, in the “Y at Cursor” column.

For a better clearness the other signals have been hidden.

Amend Computation

Rounds per minute would be a better unit for comparing the computed feedback value to the velocity feedback value. Therefore press the <Ctrl> key and click on the “Define Computation ...” button as shown above. A dialog box will be opened, here you can edit the computation.

The signal computed before must be active.

Operation of the Oscilloscope Function

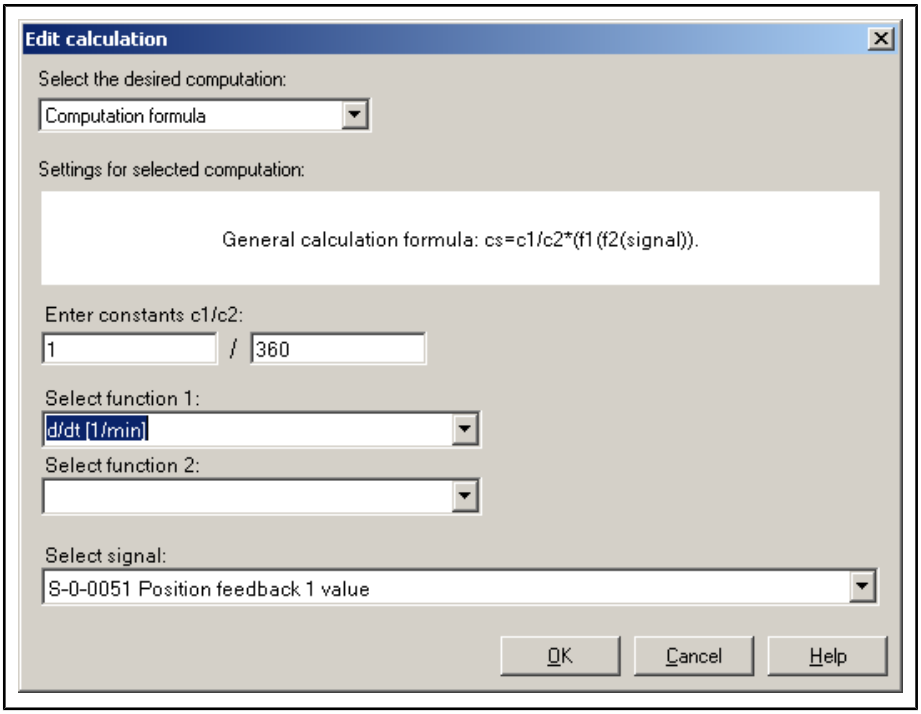


Fig.3-72: Change the settings for a computation

Select d/dt (1/min) as function 1 and repeat the computation via “OK”.

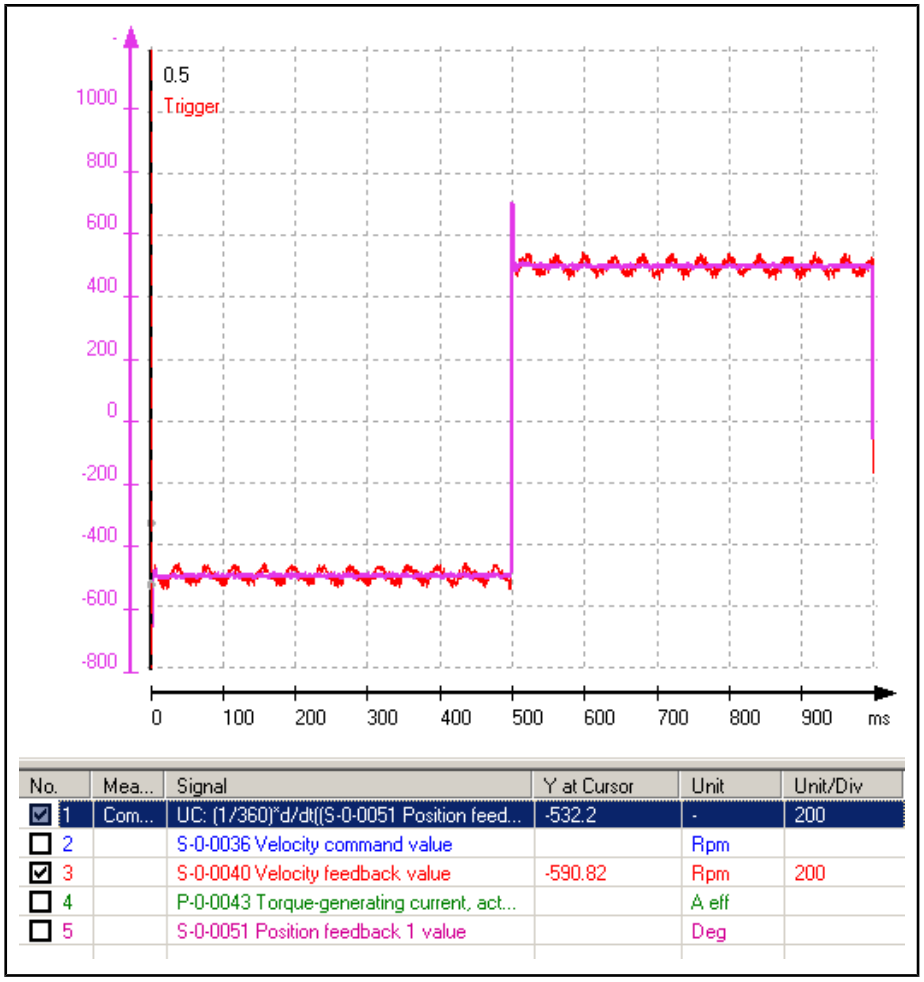


Fig.3-73: Result after derivation d/dt (1/min)

How to estimate the velocity feedback value

You can also manually obtain the velocity feedback value from the position feedback value using the two line cursors:

Operation of the Oscilloscope Function

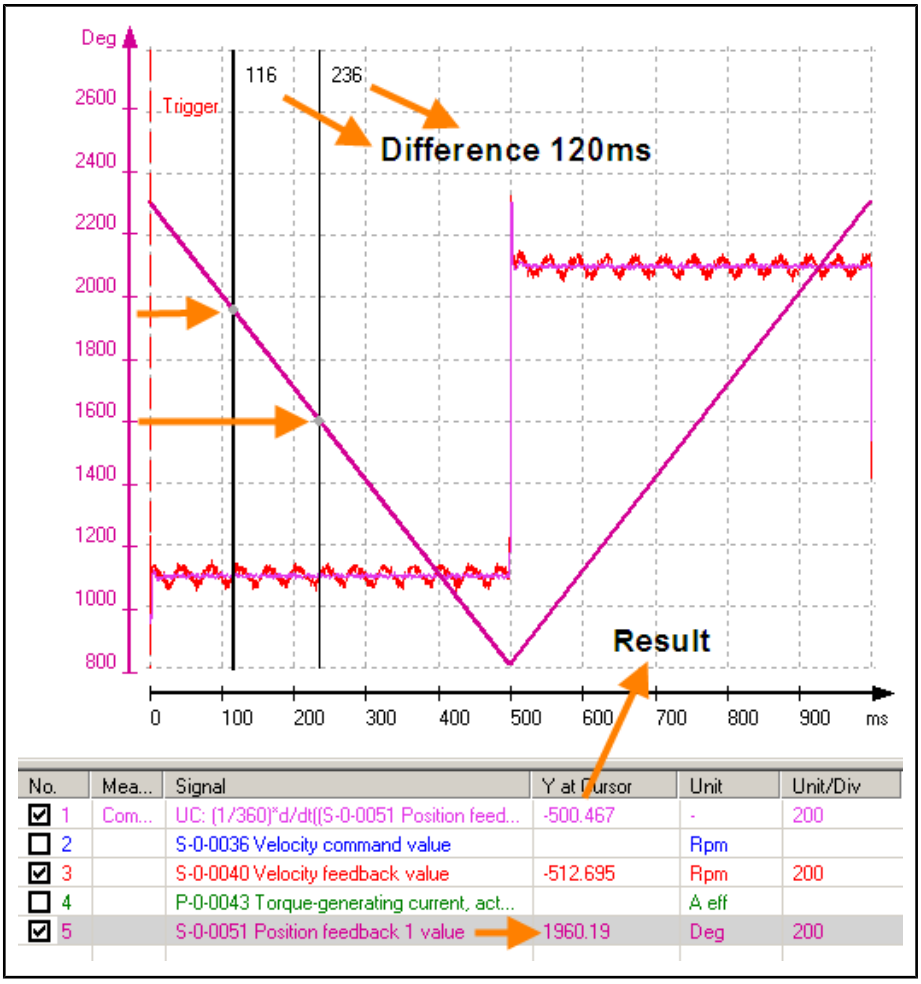


Fig.3-74: Estimation of the velocity feedback value

The curve of “S-0-0051 position feedback 1 value” has an intersection with each of the two line cursor. Place the two cursors, so that the Y-values of these intersections differ by 360° (here 1600° and 1960°). Define now the difference between the x-values of the cursors (here 236 ms - 116 ms = 120 ms).

Thus the velocity feedback value is 1 round in 120 ms, i. e., 8,333 rps or 500 rpm. This is consistent with the computed signal value in the graph.

Pre-defined Functions

The Configure Computation dialog box contains further pre-defined functions for function 1 and function 2. The figures below show the possibilities for the two functions:

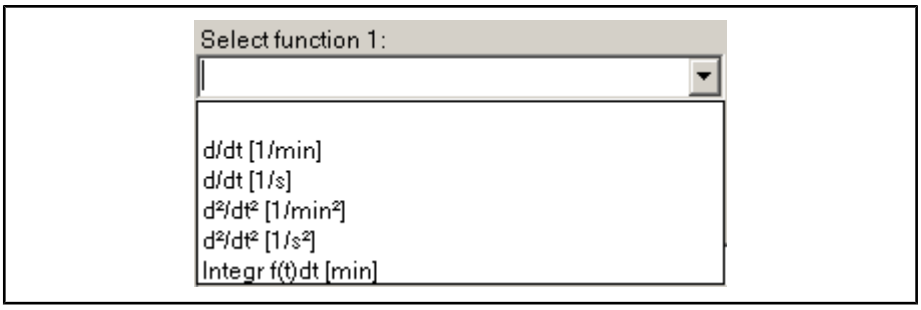


Fig.3-75: Possible selections for function 1

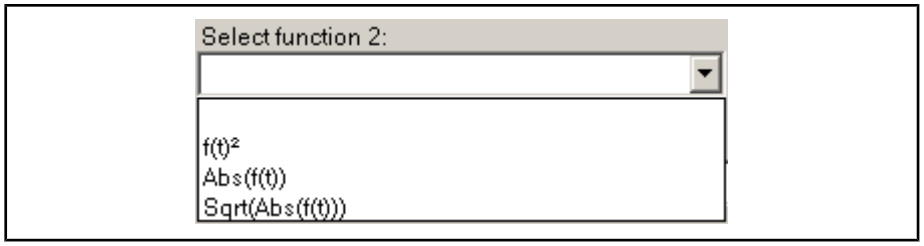


Fig.3-76: Possible selections for function 2

Thus you can, e. g., calculate the 2nd derivation from the position feedback value in order to obtain the acceleration value.

3.9.3 Summation

Example: Adding two Signals

When the computation dialog box has been started, the general computation formula is pre-selected. To add two signals, select the computation type Addition from the dialog box.

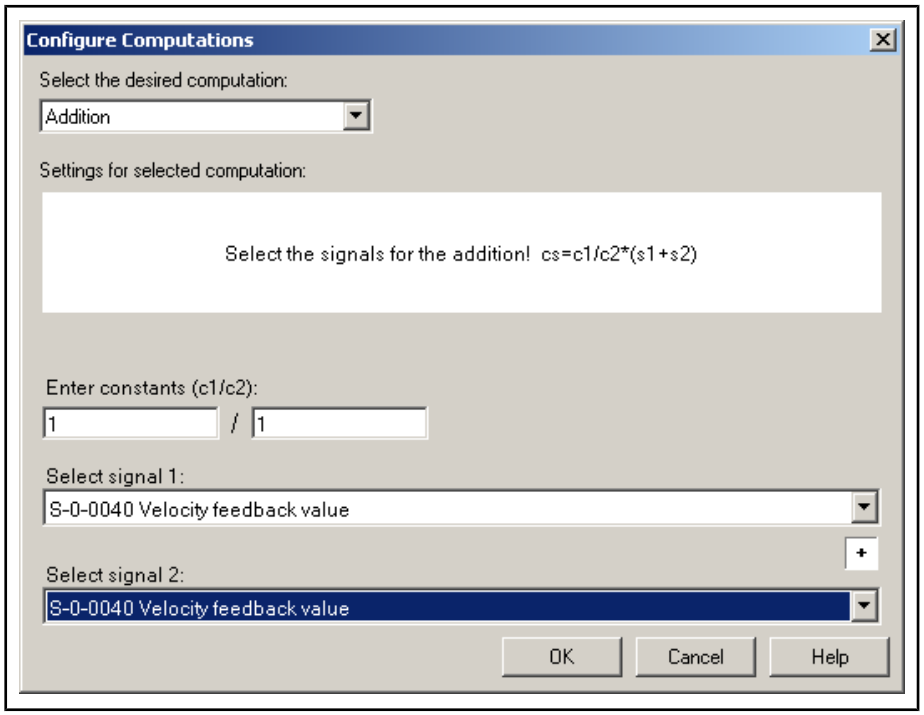


Fig.3-77: Configure Computations dialog box, addition of two signals

Now you can select two constants and two signals for the addition. In this example the constants c1 and c2 are 1. The parameter S-0-0040 has been selected as signal s1 as well as signal s2.

Click on the “OK” button to initiate the computation and to apply the computed signal to the graph and to the signal overview.

Operation of the Oscilloscope Function

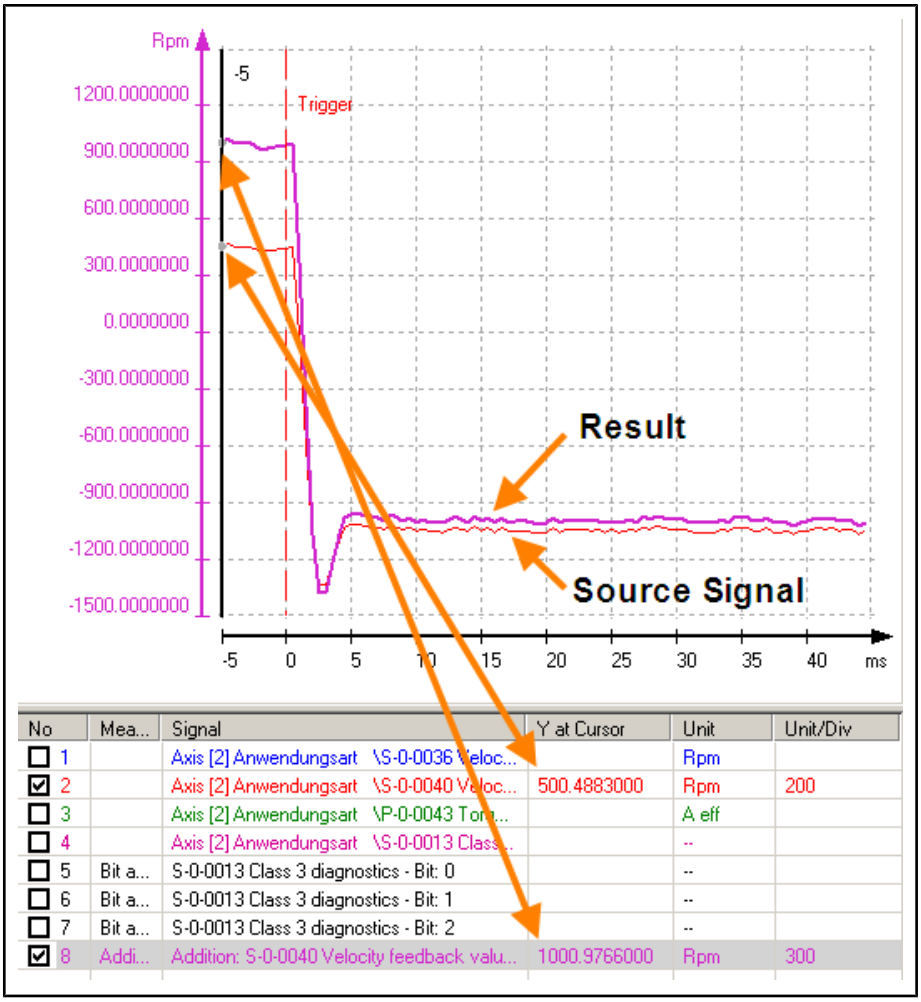


Fig.3-78: Display the sum of two signals

The values in the “Y at Cursor” column correspond with the intersections of the active cursor and the visible signals.

To discard your changes, click on “Cancel”.

3.9.4 Options for the Computation

Selecting Computations

The figure below shows a list of the available computations:

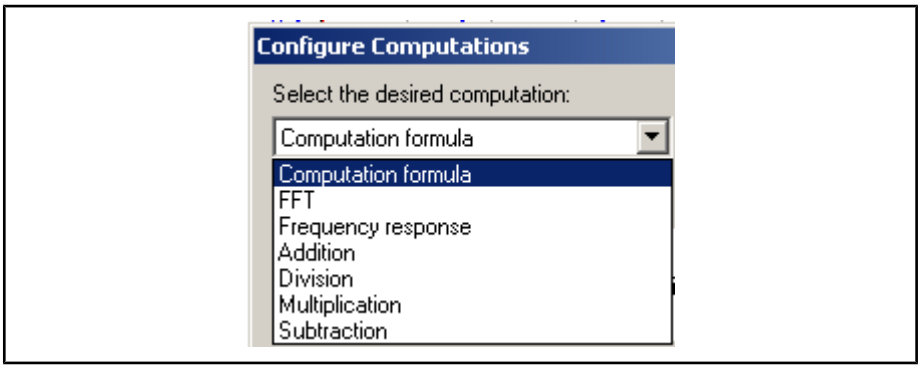


Fig.3-79: Configure Computations dialog box, available computations

Depending on the computation selected, the necessary fields are provided in the dialog box.

Fast Fourier Transformation FFT

Preparing the FFT Computation

The computation algorithm works best, if the number of measurement points is a power of 2. The following values have been selected in the example:

- 4096 measurement points
- Time grid of 0.5 ms
- 0% Pretrigger
- You have to measure the signal a full period long, at least.

After having opened the computation dialog box via “Define Computation...”, start the computation of the Fast Fourier Transformation for the selected signal by “OK”.

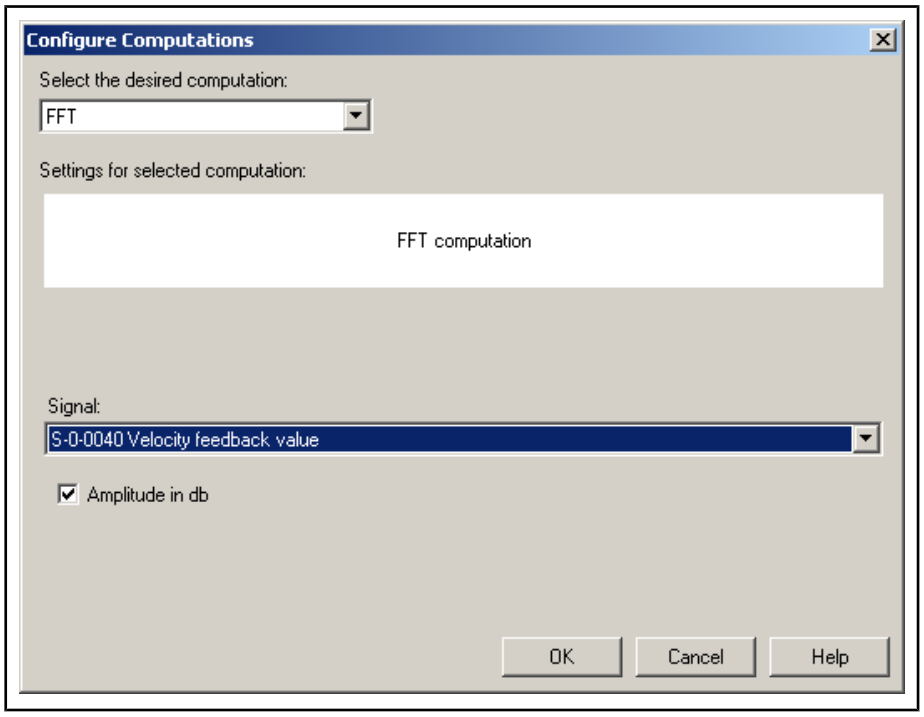


Fig.3-80: Configure Computations dialog box, FFT

You will obtain the following result, the amplitude in dB:

Operation of the Oscilloscope Function

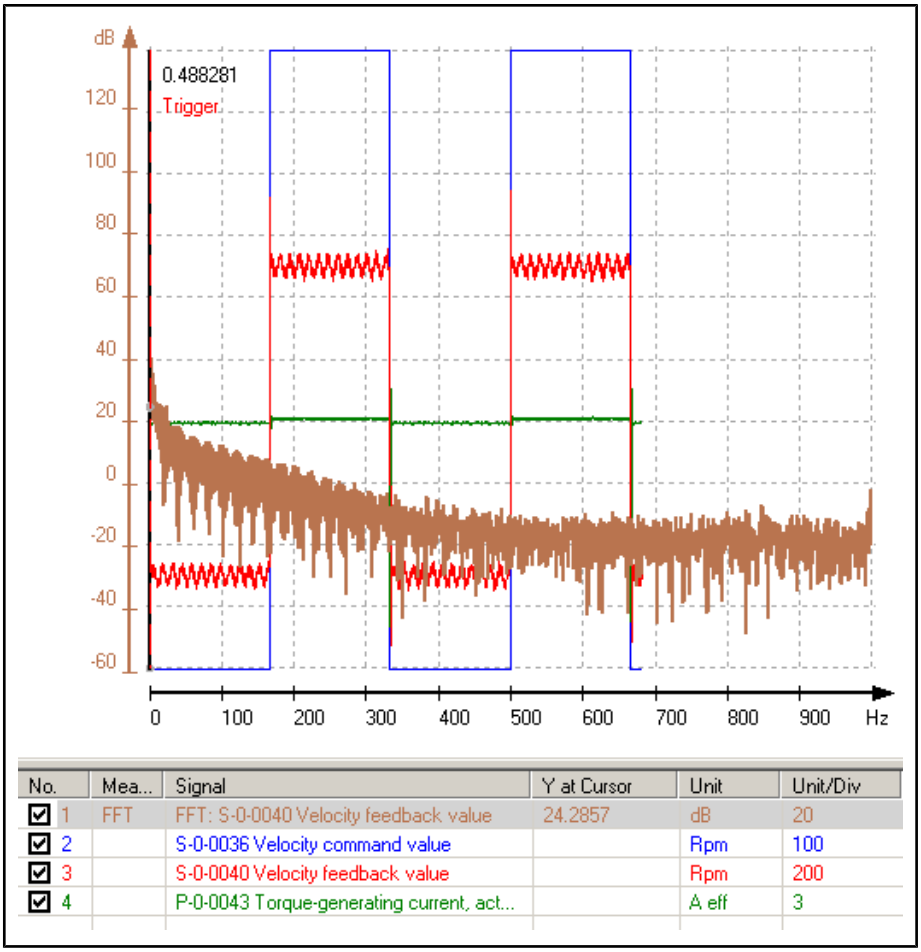


Fig.3-81: Result FFT: display in dB

Normally the amplitude of the FFT is displayed in dB, but you can also select the display according to the amplitude size. To do this, start the computation dialog box again and uncheck the “Amplitude in dB” checkbox. The figure below shows the result:

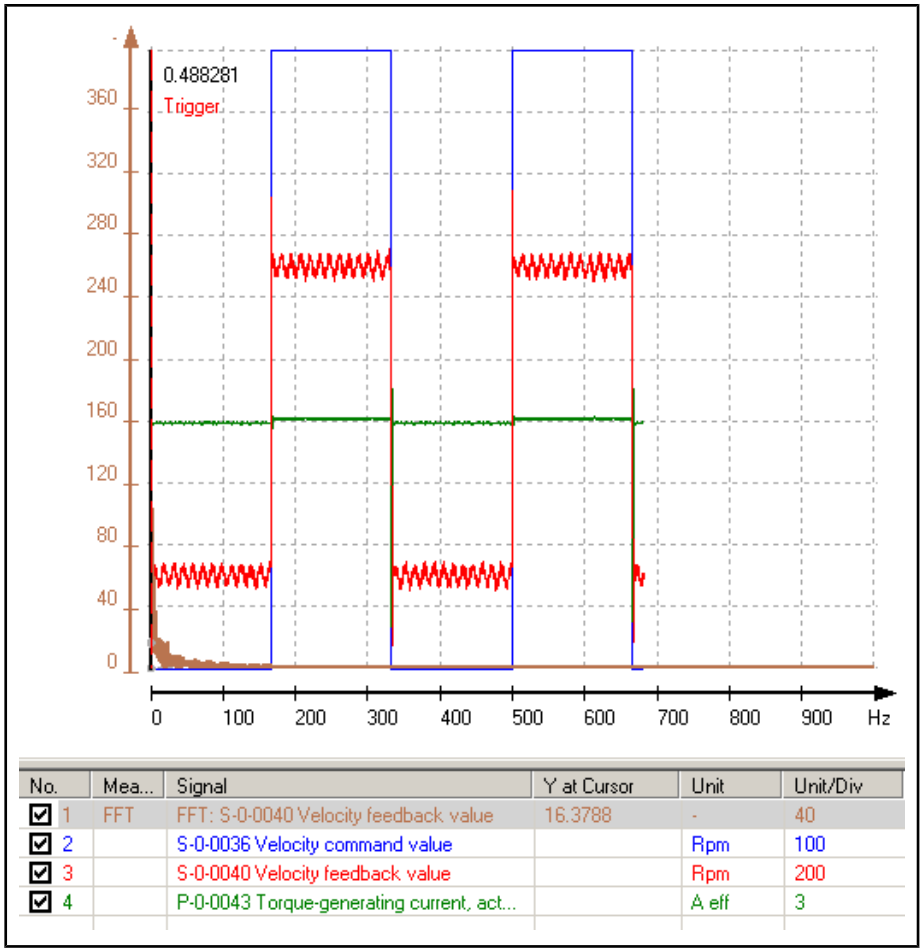


Fig.3-82: Result FFT: Display of the amplitude size

☞ Using “Ctrl” and left click on “Define Computation...” you can edit the computation selected in the signal overview.

☞ When you start the measurement again, the computation will be repeated automatically.

Frequency Response

If the frequency response computation is selected, the frequency response is computed by means of the command signal and actual signal selected.

You can compute the frequency response via the known computation dialog box or you can go to the “Frequency Response” tab page and select “Define Computation...” there.

Operation of the Oscilloscope Function

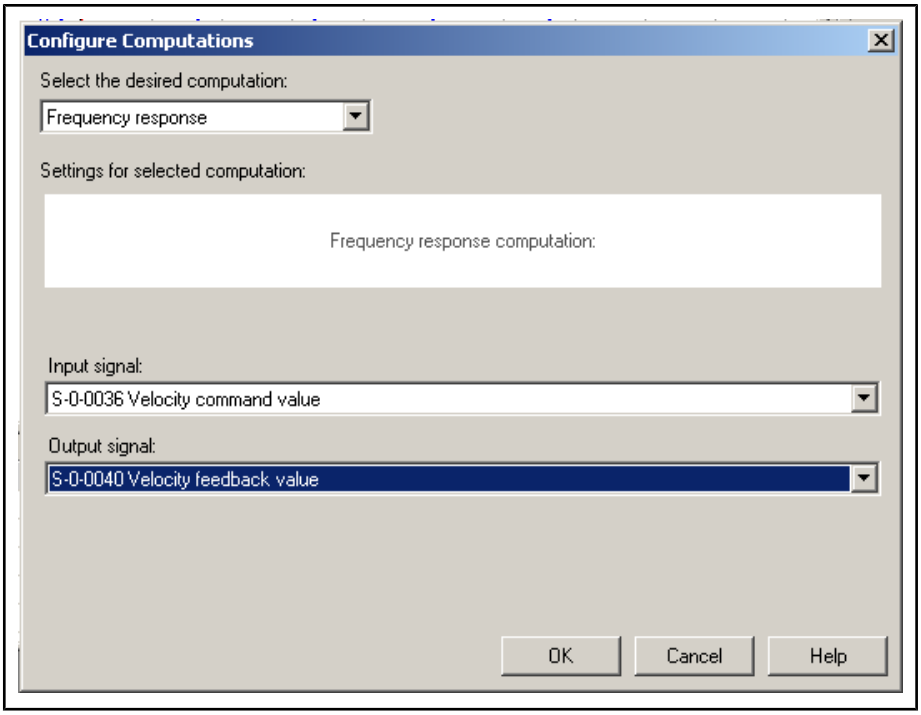


Fig.3-83: Computation dialog box: Frequency Response

The following settings were made for this example with IndraDrive:

- 4096 measurement points
- Time grid of 0.25 ms
- 0% Pretrigger
- Sending a noise signal by the drive internal noise generator to the drive
- Perform measurement
- Start the computation dialog and confirm by “OK”.

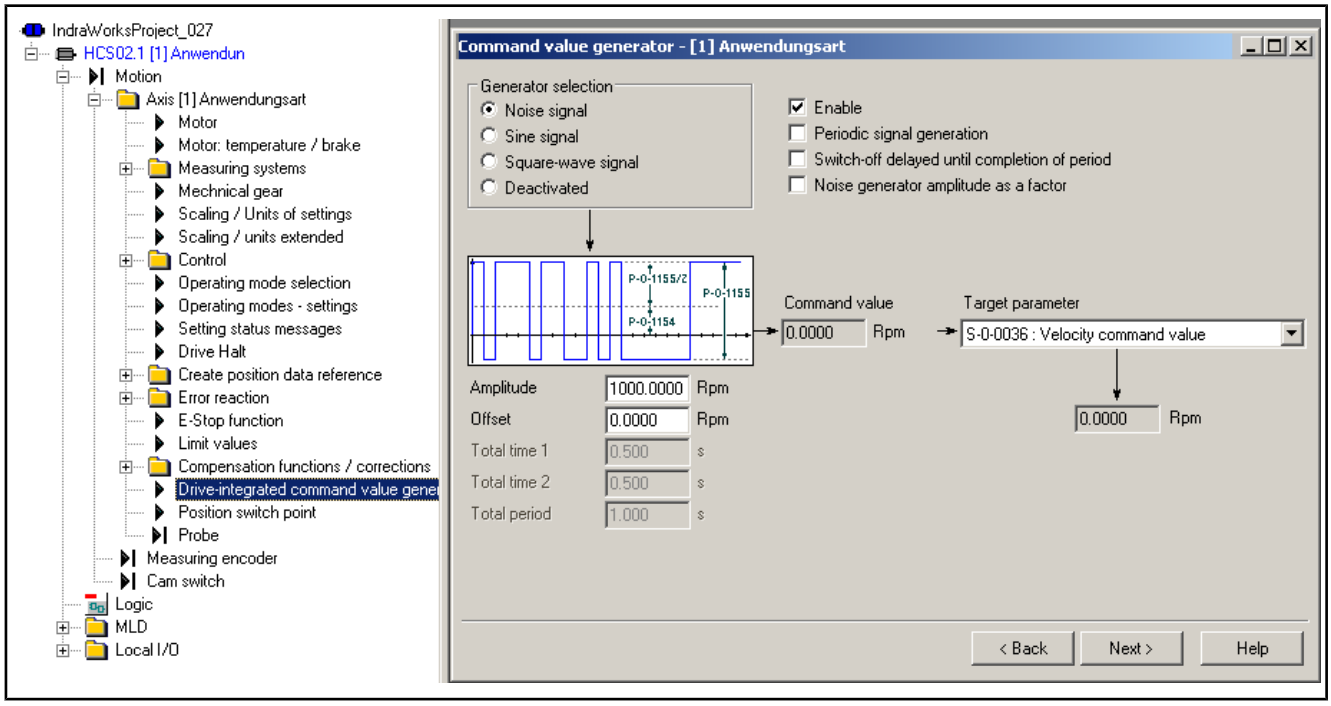


Fig.3-84: Set noise signal via drive internal command value generator

The figure below shows the result of the frequency response computation.

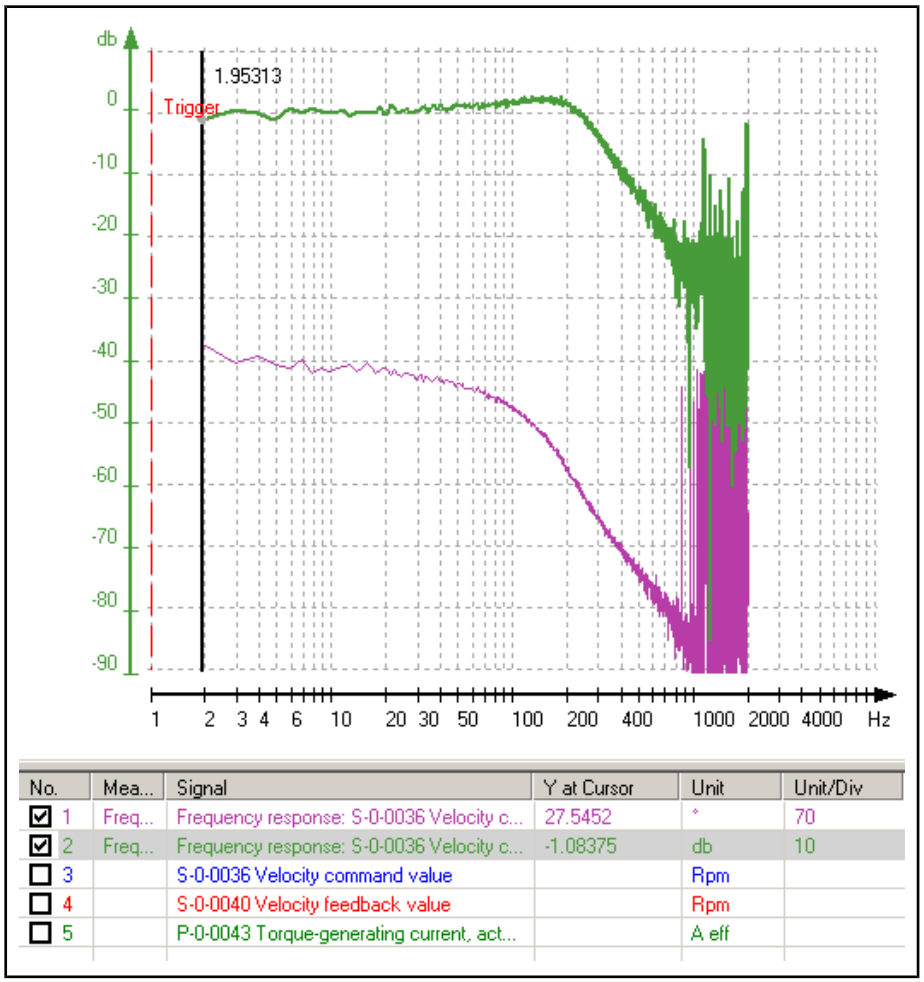


Fig.3-85: Amplitude and phase of the frequency response computation

The manual scaling has been used to enlarge the amplitude and to place the two cursors:

Operation of the Oscilloscope Function

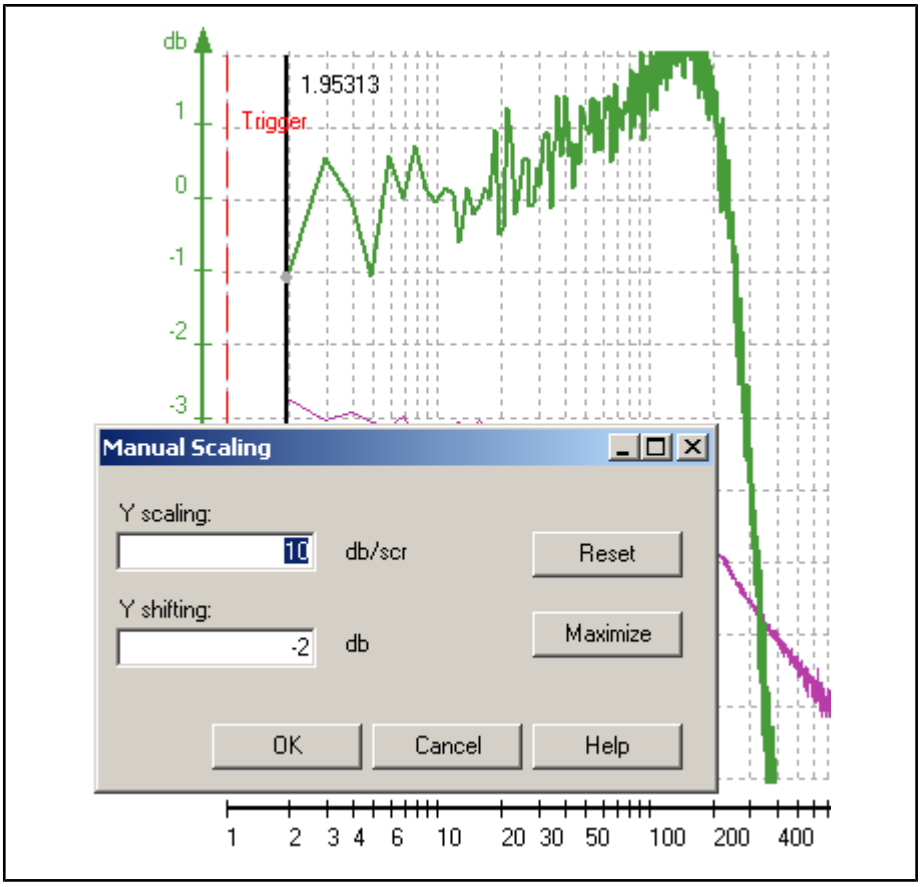


Fig.3-86: Enlarged display of amplitude

Afterwards, the area between the two cursors has been zoomed:

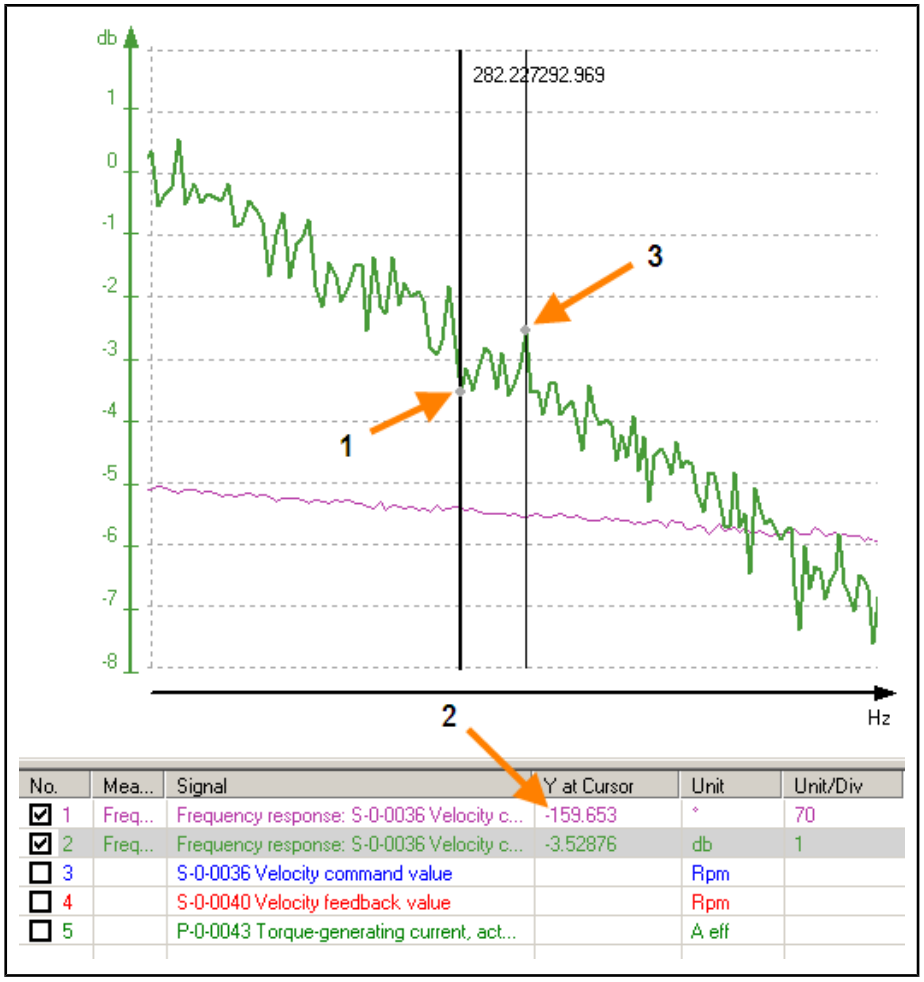


Fig.3-87: Read the -3dB limit

Example for optimization

1. Place cursor 1 at that position, where the value falls below the -3db limit for the first time.
2. Read the phase at this position(here approx. -160) and calculate the difference to -180°. The difference should be as large as possible (here approx. 20°).
3. Place cursor 2 at that position, where the value is above the -3db limit for the last time.
4. Read the phase at this position, too, and calculate the difference to -180°. The difference should be as large as possible (here approx. 5°, without illustration).

The larger the difference between the phase values and -180°, the less the system tends to natural frequencies. An optimum has to be found here.

Improving the Computation

For an improved display you can compute a moving average for each signal of the frequency response computation. To do this, use the button “Signal Smoothing” on the “Frequency Response” tab page.

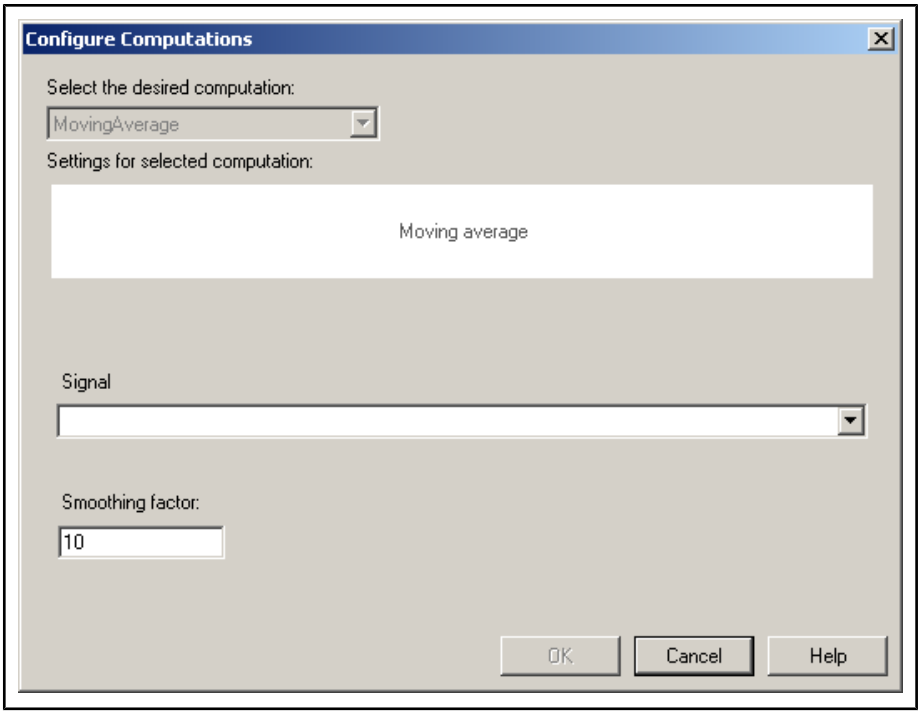


Fig.3-88: Define signal smoothing

Select the signal to be smoothed and the smoothing factor in this dialog box.

☞ A smoothing factor of 10 is preset. That means: ten consecutive values form one mean value, according to the following formula: $Y_{11\ new}=(Y_1+ .. +Y_{10}) / 10$. To compute the next mean values move the start point step by step. This procedure will provide mean values till the end of the measurement curve. The first ten values of the original measurement curve are not replaced by mean values and the original values remain in the display.

Contour Deviation

To display the contour deviation you have to measure the command positions and actual positions of the axes (only possible in MTX). The deviation will be excessively displayed in every point of the contour.

☞ Even if the display area of the graph is not a square, the contour will be displayed in a way, that the expansion in X is the same as in Y. Thus a circle will be preserved, as shown in the following example.

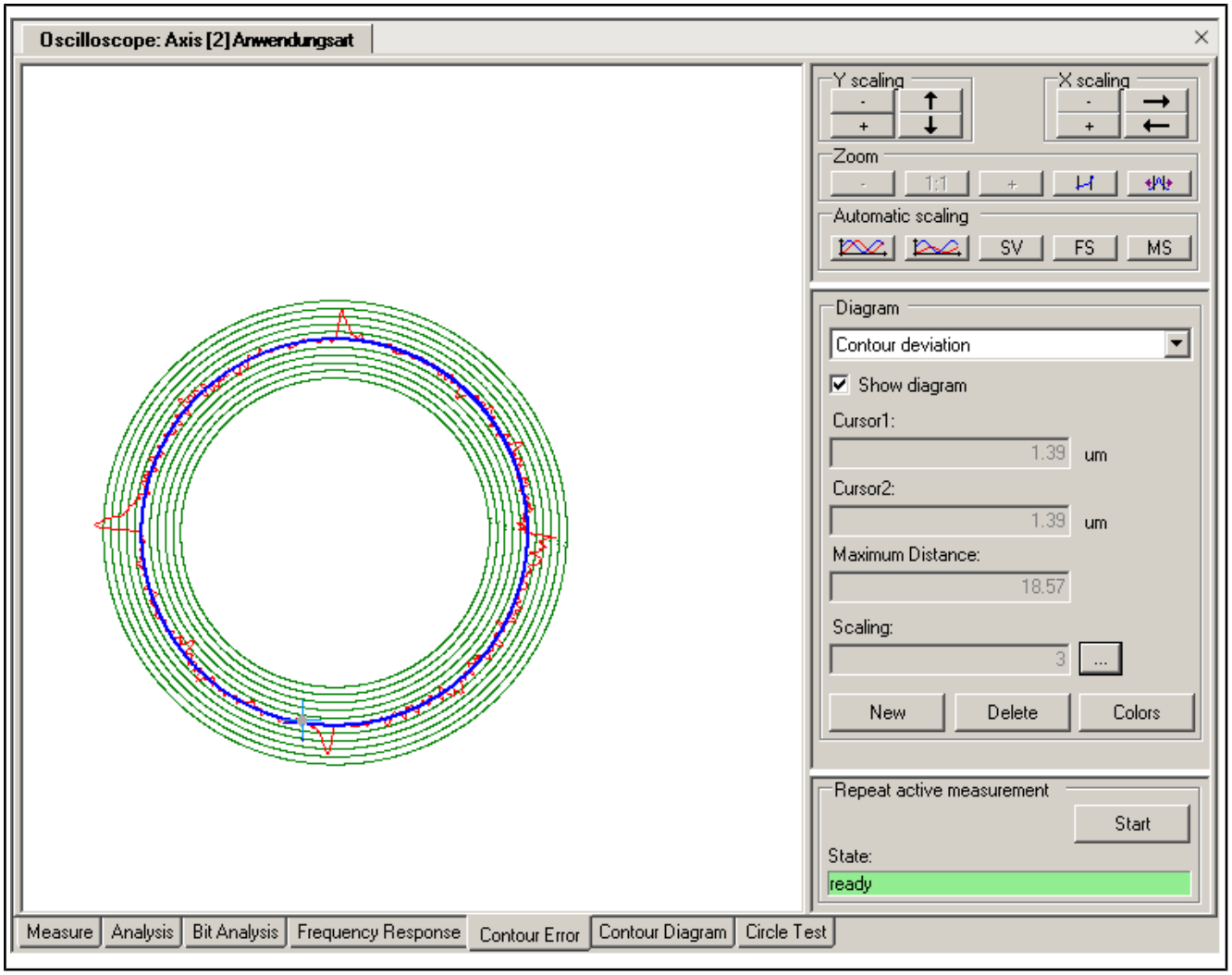


Fig.3-89: Display of contour deviation

The example shows a typical application:

The deviation when going a circle is displayed. You can see the typical errors at the quadrant crossings, caused by frictional resistance. The effect of the friction can be reduced in IndraDrive by parameters. You can check the optimization with this display.

The figure below shows an enlarged area, making the deviation clearer.

Operation of the Oscilloscope Function

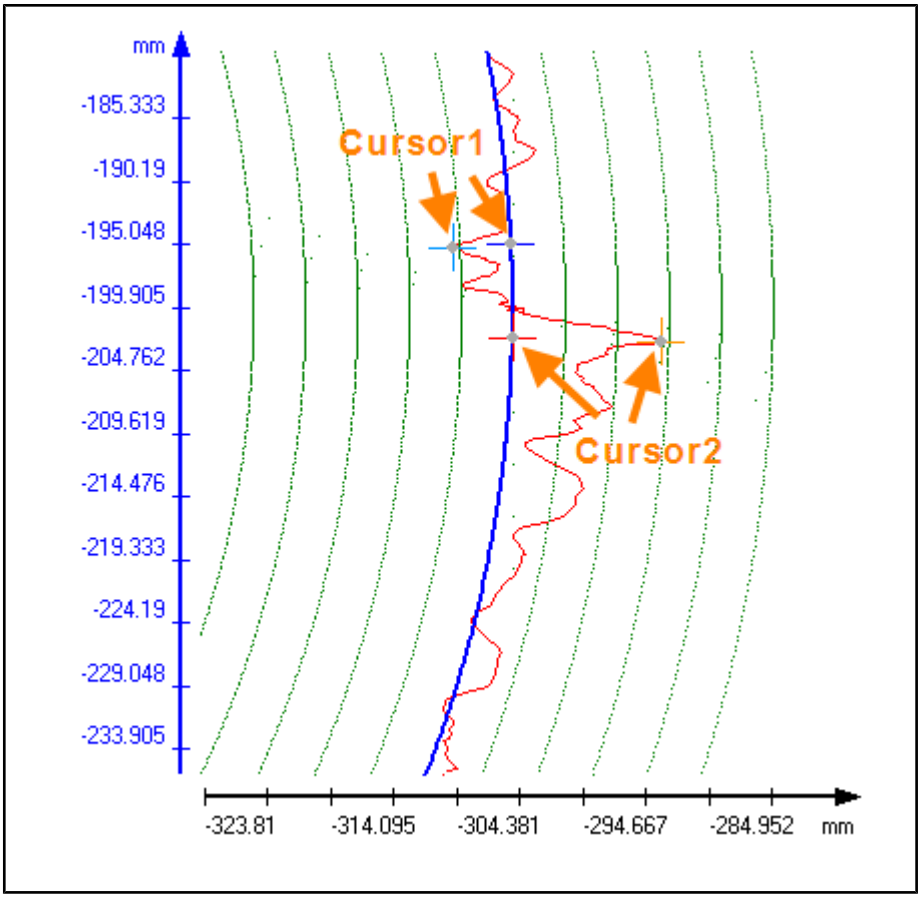


Fig.3-90: Enlarged area

Additionally, you can see the two cursors, each one at the command contour and at the actual contour.

☞ The cursor at the command contour is directly coupled to the cursor at the actual contour. For an easier understanding the contour error has been neglected.

You can show and set further options for the contour deviation:

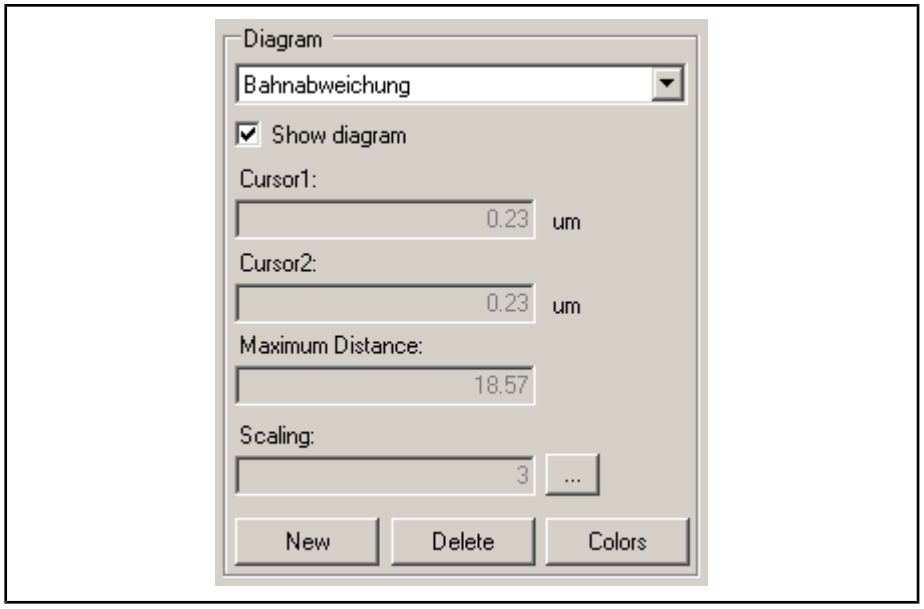


Fig.3-91: Settings for the contour deviation

Select Diagram If you have defined several diagrams, you can select one here.

Show Diagram Use this checkbox to show or hide a diagram.

Cursor1 This field displays the deviation between command contour and actual contour in µm at cursor 1.

Cursor2 This field displays the deviation between command contour and actual contour in µm at cursor 2.

Maximum Distance This field shows the maximum deviation between command contour and actual contour in µm.

Scaling This field shows the current scaling factor used to compute the amplification. It complies to the distance of two auxiliary lines.

Button "..." Set the scaling factor used to compute the amplification. The current value is displayed in the field "Scaling".

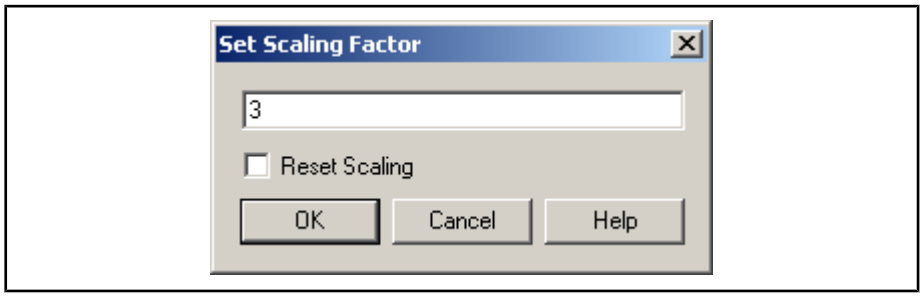


Fig.3-92: Scaling factor

Button "New" Use this button to open the computation dialog box and to select the signals for the contour display:

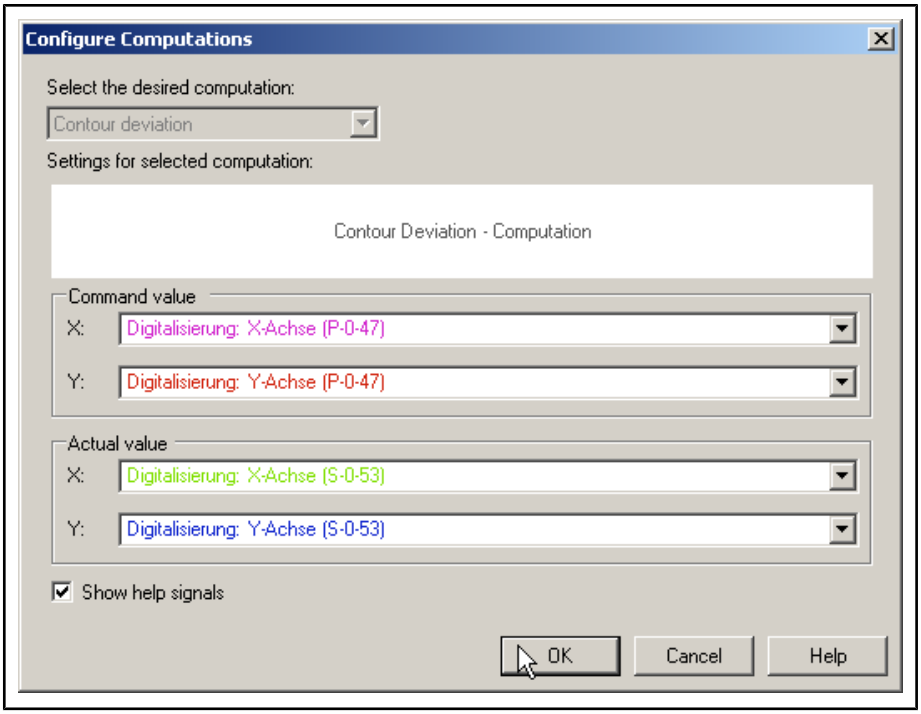


Fig.3-93: Configure Computation dialog box, select signals for contour display

Button "Delete" This button deletes the selected diagram.

Button "Colors" The colors will be assigned to the signals and subsidiary lines.

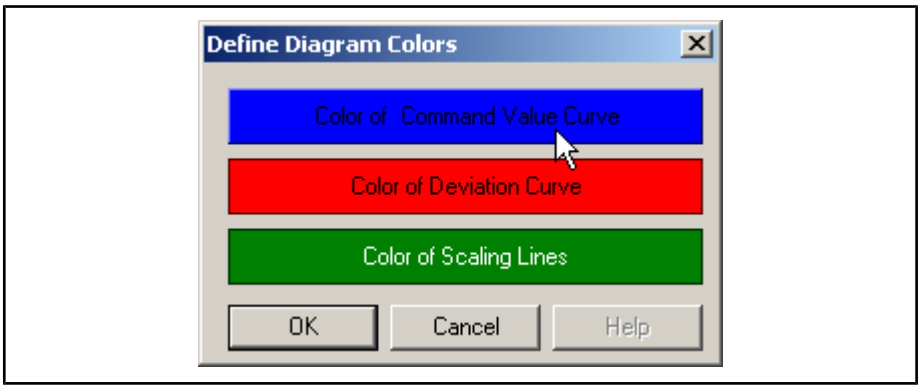


Fig.3-94: Assign the colors

Further options for editing the contour deviation:

- Scaling and moving the graph in X and Y,
- Using the zoom options,
- Automatic support for scaling,
- Repeat current measurement.

Contour Diagram

The contour diagram can combine and display two or four signals. This makes it possible to display the command contour and actual contour together as well as separately.

☞ Even if the display area of the graph is not a square, the contour will be displayed in a way, that the expansion in X is the same as in Y. Thus a circle will be preserved, as shown in the following example.

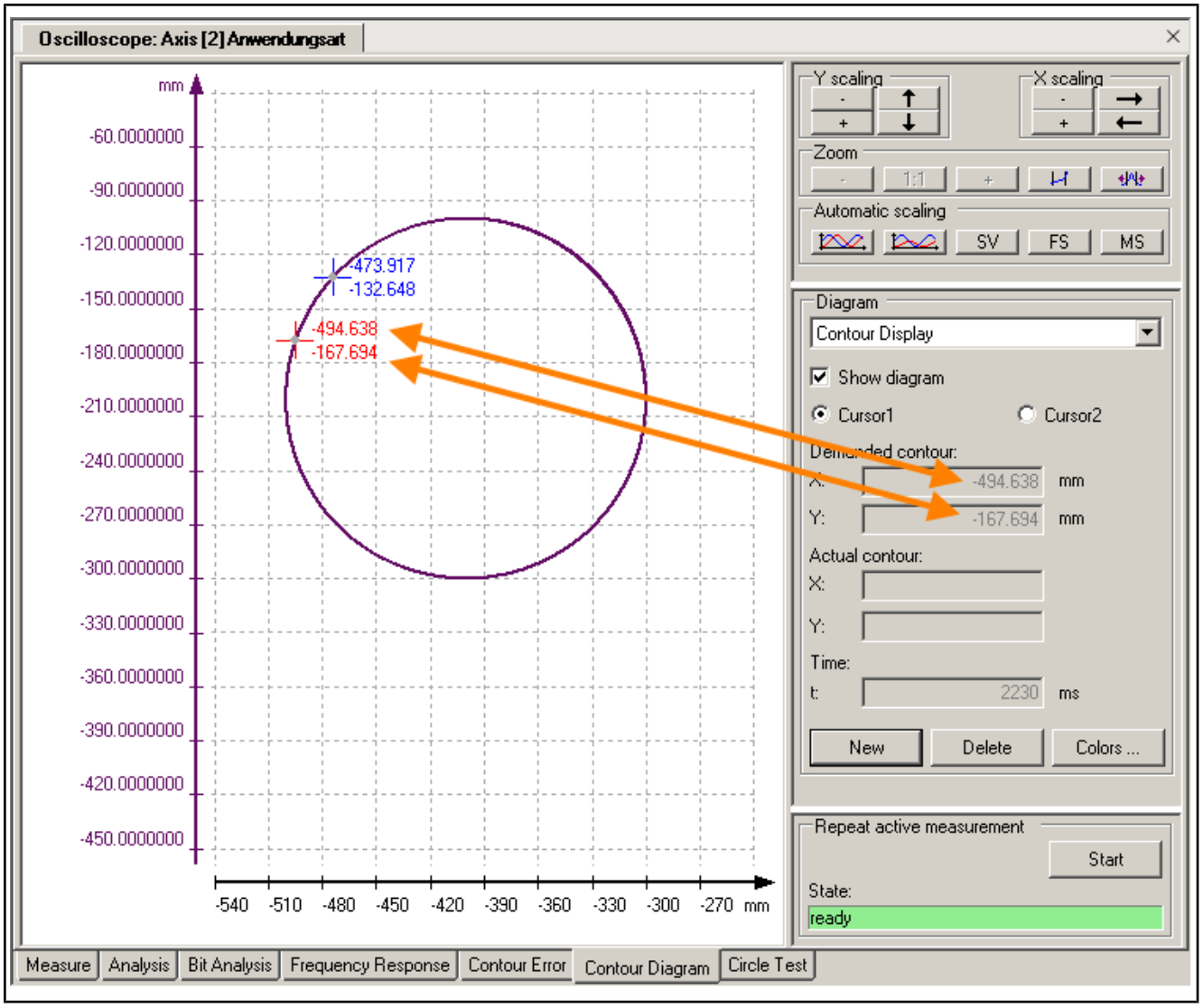


Fig.3-95: Display of the command contour

Here you can see the command contour and actual contour with the same scaling:

Operation of the Oscilloscope Function

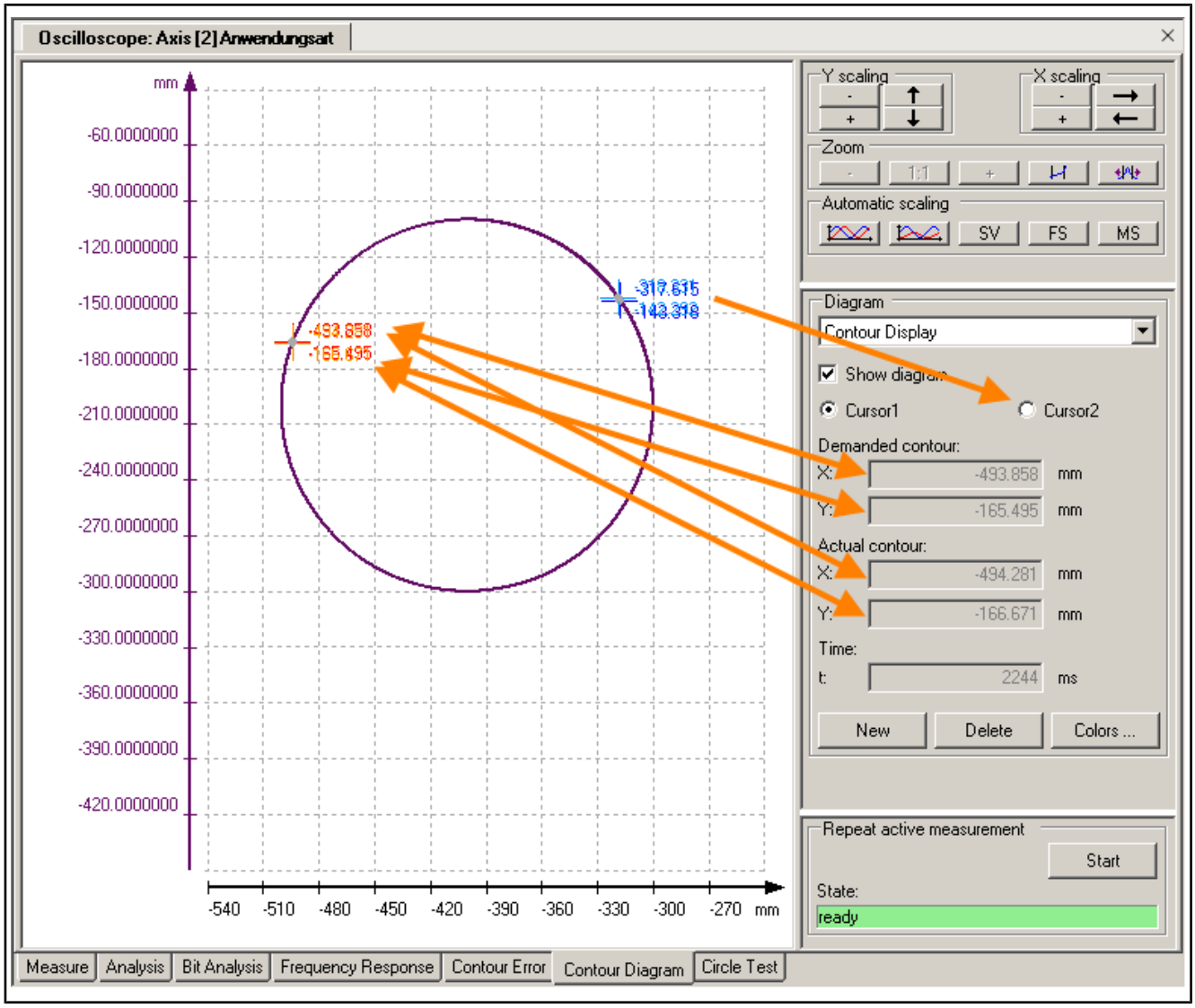


Fig.3-96: Command contour and actual contour with the same scaling

Both cursors show the x value and the y value of the command contour and of the actual contour. The values in the “Diagram” area, now representing cursor 1, correspond to the values at the cursor. If you switch to cursor 2, the current values of cursor 2 will be displayed.

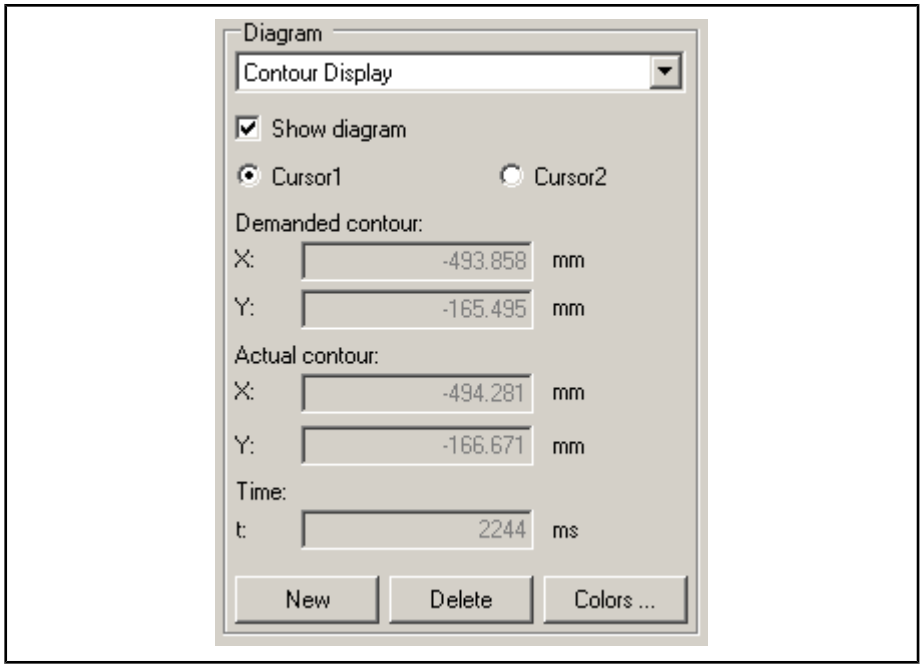


Fig.3-97: Settings of the contour diagram

Select Diagram If you have defined several diagrams, you can select one here.

Cursor1, Cursor2 Select here, whether you want to display the values of cursor 1 or cursor 2. You also define by this selection, which cursor will be moved by the cursor keys.

Command Contour X value and y value of the active cursor

Actual Contour X value and y value of the active cursor

Time Recording time of the measurement, starting at 0 in the first quadrant.

Button “New” Use this button to open the computation dialog box and to select the signals for the contour display:

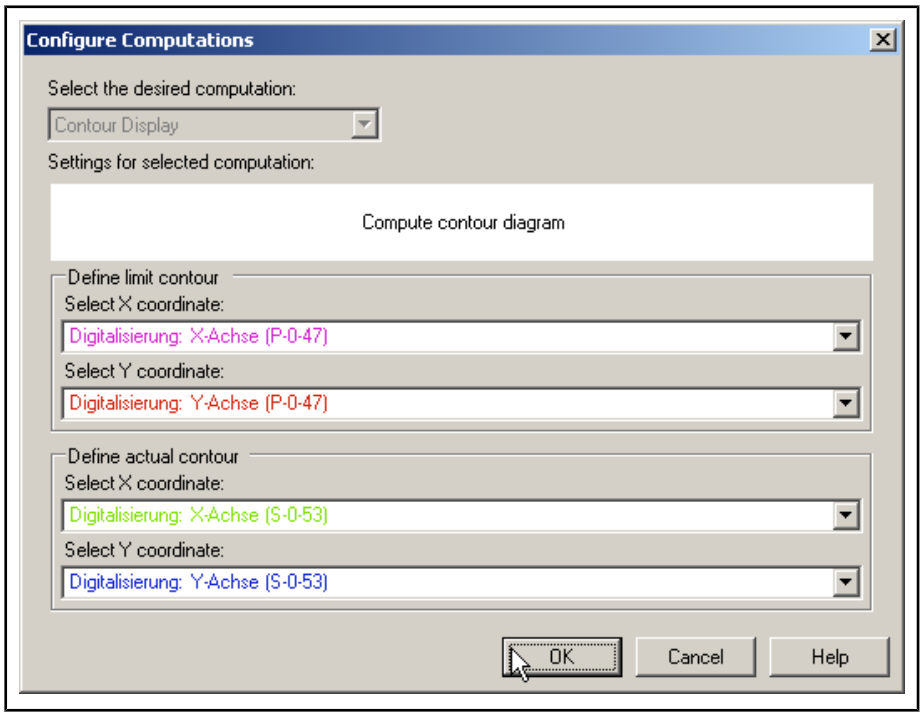


Fig.3-98: Configure Computation dialog box, select signals for contour display

Button “Delete” This button deletes the selected diagram.

Button “Colors...” The colors will be assigned to the signals and subsidiary lines.

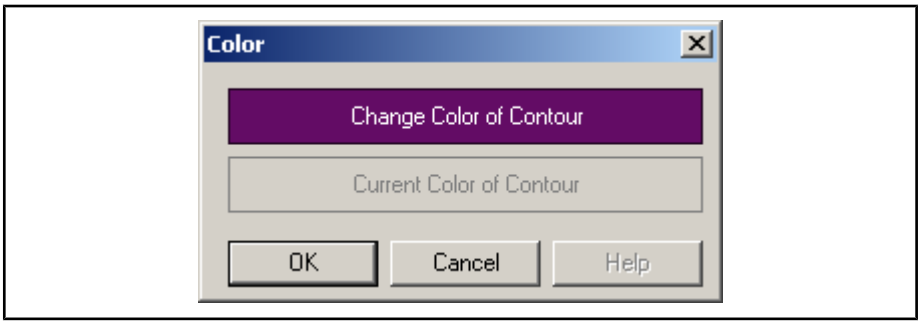


Fig.3-99: Assign the colors

Circle Test

To perform a circle test you will need an MTX with two drives.

The circle test acts on the following conditions and contains the following functionalities:

- You have to create an NC test program to drive two axes, requesting a circular motion.
- Set the program trigger as trigger condition with
  - the name of the CNC program,
  - the program line to be triggered,
  - the channel number.
- Start the measurement.
- Start the NC test program.
- If the cirular motion has been completed, the measurement results will be displayed.
- The measurement results can be labeled and stored separately.
- You can also view old measurements and compare them with the new measurement results.

See the following example for a circle test:

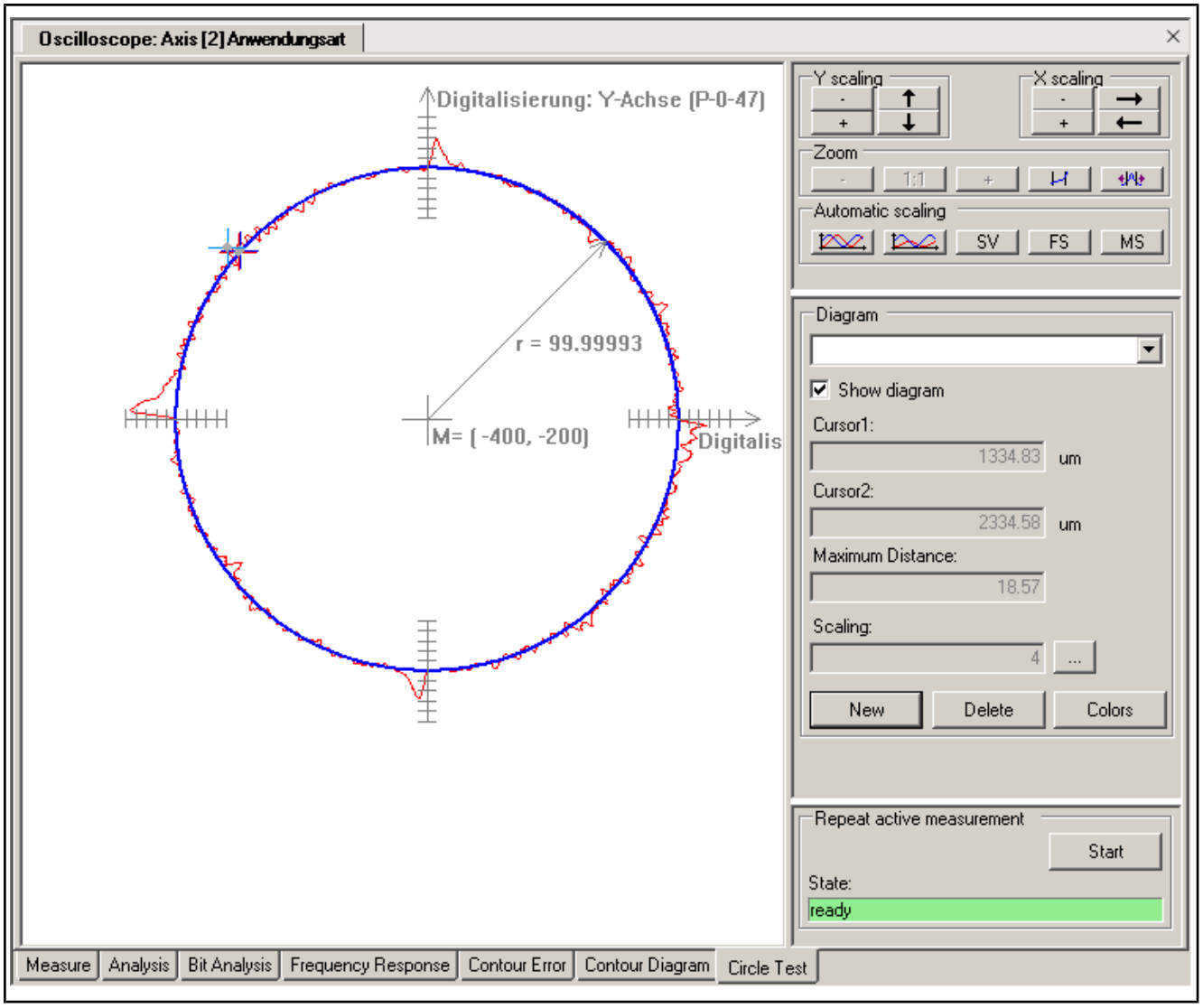


Fig.3-100: Result of the circle test

The circle test assumes, that a circle contour has been measured.

It uses the contour deviation formula to compute the actual contour deviation.

3.10 Loading and Saving a Measurement

3.10.1 General Information

Use the **Oscilloscope** menu to load configuration data and measurement data via import, and to save them via export.

Depending on the menu item the following file formats are supported:

ZIP Compressed file in ZIP format, containing a complete measurement.

XML Text file in XML format, containing a complete measurement.

OCD Text file in XML format, containing the configuration of a measurement.

TXT Output of the measured data as text file which can be read in Excel.

The following menu items are provided:

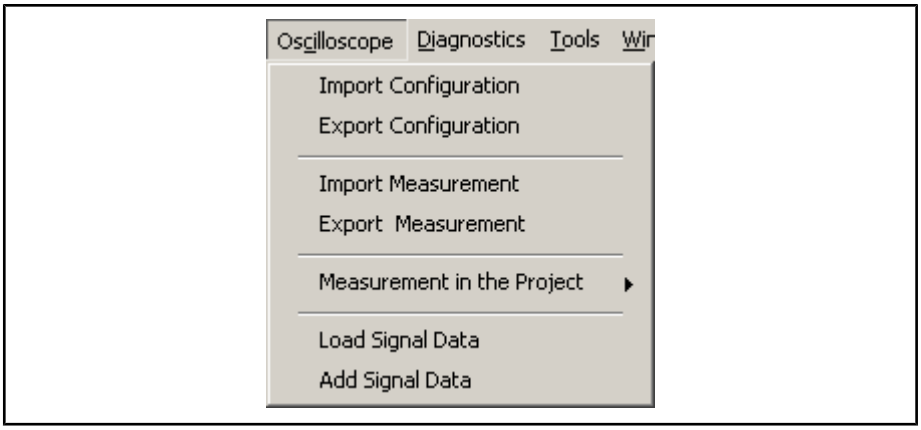


Fig.3-101: Menu items for the oscilloscope

3.10.2 Import Configuration, Export Configuration

Only the configuration of a measurement will be read or saved in a separate file. The file extension is OCD.

3.10.3 Import Measurement, Export Measurement

A measurement will be loaded from an arbitrary directory or stored in an arbitrary directory. The files with the extension ZIP, XML and OCD can be loaded The files with the extension ZIP, XML and TXT can be stored.

3.10.4 Measurement in the Project Submenu

General Information

The **Measurement in the Project** sub menu offers items for the management of the measured data in the IndraWorks project directory. Only zip files will be supported.

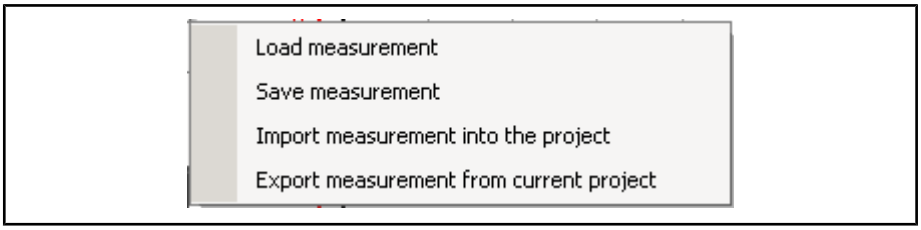


Fig.3-102: Measurement in Project Submenu

Load Measurement

Load Measurement loads a measurement into the oscilloscope. This measurement consists of the measurement configuration, possible computation configurations, and the signal data. The signal data will be shown in the graph. The configuration data will be saved internally. These data can be displayed, changed and sent to the device for a restart of the measurement, using the according dialogs (for signals, measurement, trigger, computations).

Use this menu entry to load and display a measurement stored in the current project. A previously displayed measurement will be removed from the display. Select a measurement in the following dialog box:

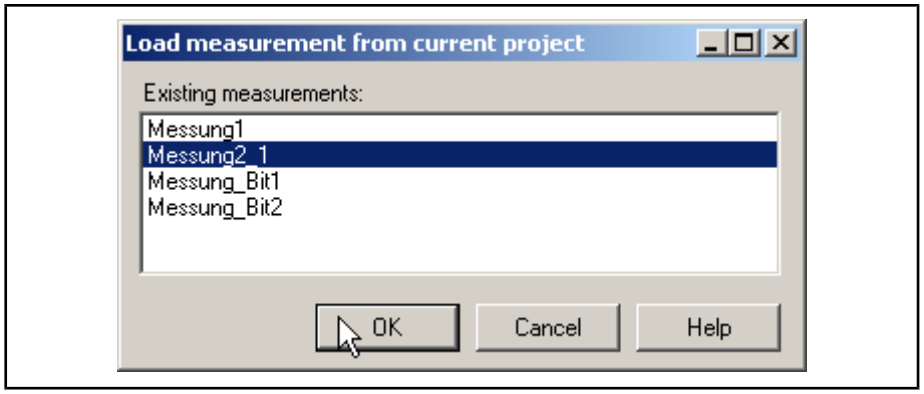


Fig.3-103: Load Measurement from Current Project dialog box

Save Measurement

Save Measurement stores a measurement together with its measurement configuration, a possible computation configuration and its signal data.

Use this menu entry to save the current measurement in the current project directory. Specify the new name of the measurement in the following dialog box:

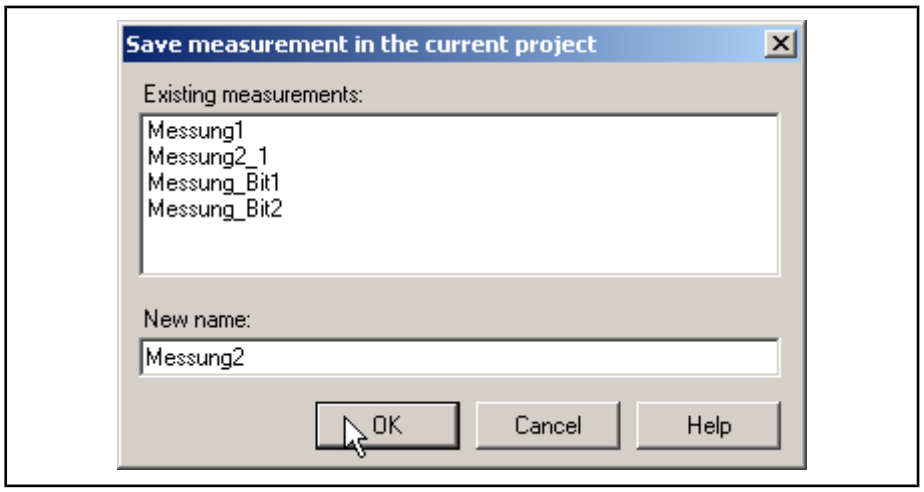


Fig.3-104: Save Measurement in Current Project dialog box

Import Measurement into the Project

Use this menu entry to copy a measurement from an arbitrary directory into the project without displaying it automatically. If you want to display the measurement use **Load Measurement**.

Export Measurement from Current Project

Use this menu entry to copy measurements from the project directory into an arbitrary directory. The following dialog box will be opened:

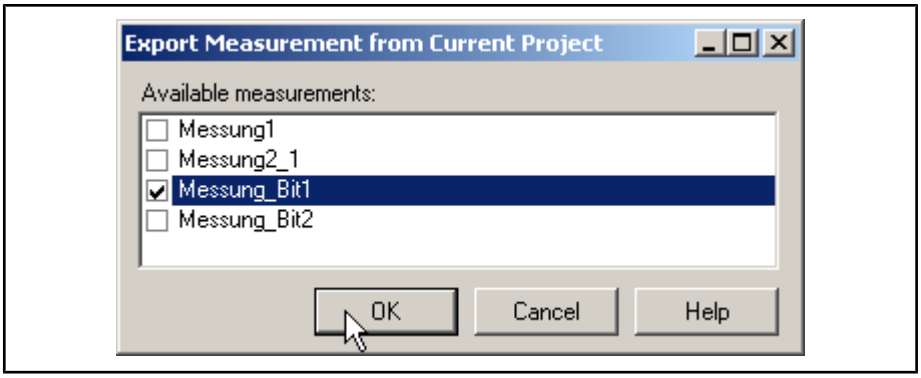


Fig.3-105: Export Measurement from Current Project dialog box

Select the measurement to be copied and click on “Next”. Select the destination directory for the measurements or create a new one in the following dialog box.

3.11 Document Window in Clipboard

Press <Ctrl> <Alt> <J> to copy the entire workspace of the oscilloscope (graph with control elements) as screenshot to the clipboard. Thus the displayed measurement can be applied to other programs for documentation purposes.

3.12 Exporting a Measurement as Text File

Press <Ctrl> <Alt> <R> to copy a measurement as text to the clipboard. This allows you to apply the values directly to Excel, without having to use a separate file of the **Export Measurement** menu.

3.13 References

Please use the following references for support on signals which are suitable for being recorded graphically.

- Rexroth IndraDrive Drive Controllers, Parameter Description; DOK-INDRV*-GEN-**VRS**-PA02-EN-P
- Rexroth IndraDrive, Firmware for Drive Controllers MPH-02, MPB-02, MPD-02 Functional Description DOK-INDRV*-MP*-**VRS**-FK01-EN-P
- Rexroth IndraDrive Firmware for Drive Controllers MPx02, MPx02 and HMV, Troubleshooting Guide; DOK-INDRV*-DEN-**VRS**-WA02-EN-P

Index

Index

Index

Notes

Rexroth
Bosch Group

Bosch Rexroth AG
Electric Drives and Controls
P.O. Box 13 57
97803 Lohr, Germany
Bgm.-Dr.-Nebel-Str. 2
97816 Lohr, Germany
Phone +49 (0)93 52-40-50 60
Fax +49 (0)93 52-40-49 41
service.svc@boschrexroth.de
www.boschrexroth.com

R911320741

Printed in Germany
DOK-IWORKS-ENGINEE*V07-AW01-EN-P