KUKA

KUKA System Technology
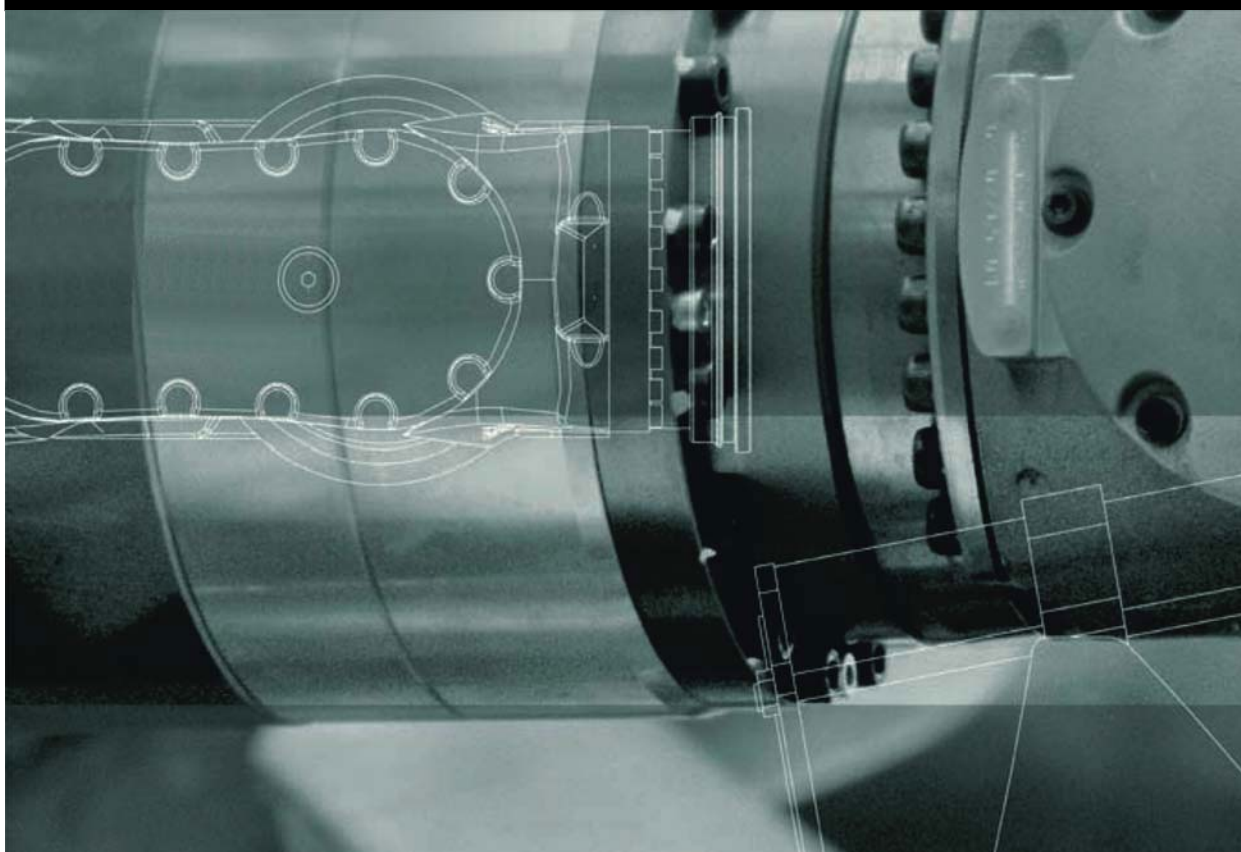
KUKA Roboter GmbH

WorkVisual 4.0

For KUKA System Software 8.2, 8.3 and 8.4

For VW System Software 8.2 and 8.3

Issued: 29.05.2015

Version: KST WorkVisual 4.0 V3

Contents

KUKA

1 Introduction

1.1 Target group

This documentation is aimed at users with the following knowledge and skills:

- Advanced knowledge of the robot controller system
- Advanced knowledge of bus technologies

> **i** For optimal use of our products, we recommend that our customers take part in a course of training at KUKA College. Information about the training program can be found at www.kuka.com or can be obtained directly from our subsidiaries.

1.2 Representation of warnings and notes

Safety

These warnings are relevant to safety and **must** be observed.

> **⚠ DANGER** These warnings mean that it is certain or highly probable that death or severe injuries **will** occur, if no precautions are taken.

> **⚠ WARNING** These warnings mean that death or severe injuries **may** occur, if no precautions are taken.

> **⚠ CAUTION** These warnings mean that minor injuries **may** occur, if no precautions are taken.

> **NOTICE** These warnings mean that damage to property **may** occur, if no precautions are taken.

> **⚠** These warnings contain references to safety-relevant information or general safety measures.
> These warnings do not refer to individual hazards or individual precautionary measures.

This warning draws attention to procedures which serve to prevent or remedy emergencies or malfunctions:

> **SAFETY INSTRUCTIONS** Procedures marked with this warning **must** be followed exactly.

Hints

These notices serve to make your work easier or contain references to further information.

> **i** Tip to make your work easier or reference to further information.

1.3 Trademarks

Windows is a trademark of Microsoft Corporation.

Pentium is a trademark of Intel Corporation.

Step 7 is a trademark of Siemens AG.

PC WORX is a trademark of Phoenix Contact.

1.4 Licenses

This KUKA software product uses open-source software. The licensing terms are displayed during installation of the KUKA software product.

1.5 Terms used

Term	Description
CK	Customer-built Kinematics
DTM	Device Type Manager
KCP	KUKA Control Panel General name for KUKA teach pendants
KLI	KUKA Line Interface The KLI is the Ethernet interface of the robot controller for external communication. It is a physical interface and can contain multiple virtual interfaces. The KLI is configured in the KUKA System Software.
KRL	KUKA robot programming language (KUKA Robot Language)
KSI	KUKA Service Interface Interface on the CSP on the control cabinet The WorkVisual PC can either connect to the robot controller via the KLI or it can be plugged into the KSI.
KSS	KUKA System Software
KUKA smartHMI	Name of the graphical user interface for the (V)KR C4 robot controller
KUKA smartPAD	Name of the KCP for the (V)KR C4 robot controller
OPS	Offline Programming System Also called "Office PC".
Safety options	Generic term for options which make it possible to configure additional safe monitoring functions in addition to the standard safety functions. Example: SafeOperation

2 Product description

2.1 Overview of WorkVisual

The **WorkVisual** software package is the engineering environment for KR C4 controlled robotic cells. It offers the following functionalities:

- Configuring and connecting field buses
- Programming robots offline
- Configuring machine data
- Configuring RoboTeams offline
- Editing the safety configuration
- Editing TOOL and BASE coordinate systems
- Defining robot cells online
- Transferring projects to the robot controller
- Loading projects from the robot controller
- Comparing a project with another project and accepting differences where necessary
- Managing long texts
- Managing option packages
- Diagnostic functionality
- Online display of system information about the robot controller
- Configuring traces, starting recordings, evaluating traces (with the oscilloscope)
- Editing the file system of the robot controller online
- Program debugging
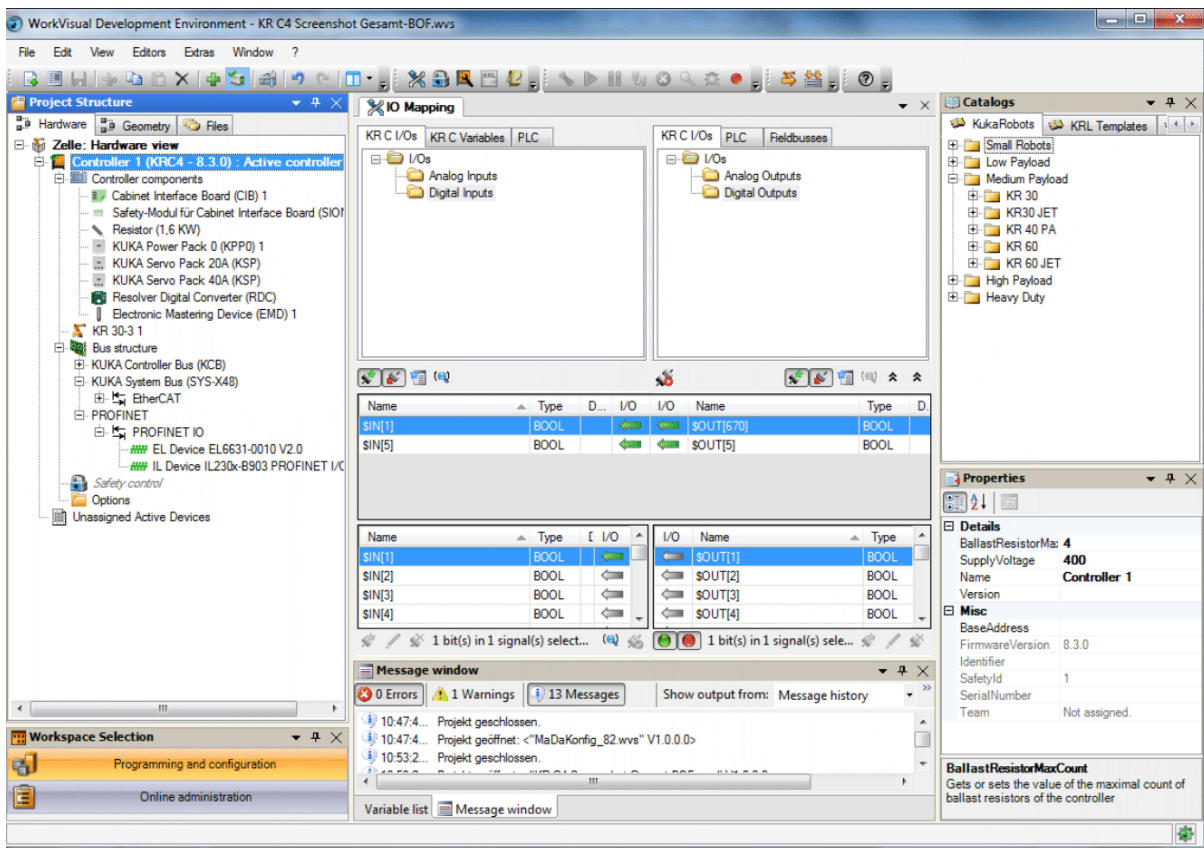
Fig. 2-1: WorkVisual graphical user interface

2.2 Intended use of WorkVisual

Use

The WorkVisual software package is intended exclusively for the configuration, programming and diagnosis of a KUKA industrial robot or customer-specific kinematic system.

Each version of the WorkVisual software package may be operated exclusively in accordance with the specified system requirements.

Misuse

Any use or application deviating from the intended use is deemed to be impermissible misuse. KUKA Roboter GmbH is not liable for any damage resulting from such misuse. The risk lies entirely with the user.

Examples of such misuse include:

- Configuration of a kinematic system that is neither a KUKA industrial robot nor a customer-specific kinematic system
- Operation of WorkVisual not in accordance with the specified system requirements

3 Safety

This documentation contains safety instructions which refer specifically to the software described here. The fundamental safety information for the industrial robot can be found in the "Safety" chapter of the Operating and Programming Instructions for System Integrators or the Operating and Programming Instructions for End Users.

⚠ The "Safety" chapter in the operating and programming instructions of the KUKA System Software (KSS) must be observed. Death to persons, severe injuries or considerable damage to property may otherwise result.

⚠ **WARNING** WorkVisual can be used with write access to modify outputs in the robot controller, without this being noticed by any persons located inside the system.
In the test modes, such access is not permitted. Outputs of the robot controller must not be modified using WorkVisual ("Single Point of Control" principle)! Death to persons, severe injuries or considerable damage to property may otherwise result.

⚠ **WARNING** WorkVisual can be used with write access to modify programs, I/O assignments, signal declarations and other parameters in the robot controller. The following applies here:
New or modified programs must always be tested first in Manual Reduced Velocity mode (T1).
After modifications to the industrial robot, existing programs must always be tested first in Manual Reduced Velocity mode (T1). This applies to all components of the industrial robot and includes modifications to the software and configuration settings. In particular, this also applies when a WorkVisual project has been activated on the robot controller.

⚠ **WARNING** After importing a safety configuration or parts thereof, the safety configuration must be checked! If this is not done, this can lead to the possibility of the robot being operated with incorrect data when the project is transferred to the real robot controller. Death to persons, severe injuries or considerable damage to property may result.

⚠ **WARNING** When activating a project on the KUKA smartHMI, an overview is displayed of the changes which will be made in comparison to the project that is still active on the robot controller.
If changes are listed in the overview under the heading **Safety-relevant communication parameters**, this means that the behavior of the Emergency Stop and "Operator safety" signal may have changed compared with the previous project.
After activation of the project, the Emergency Stop and the "Operator safety" signal must be checked for safe functioning. If the project is activated on several robot controllers, this check must be carried out for every robot controller. Failure to carry out this check may result in death to persons, severe injuries or considerable damage to property.

⚠ **WARNING** After activation of a project on the robot controller, the safety configuration must be checked there! If this is not done, the robot will possibly be operated with incorrect data. Death, injuries or damage to property may result.
 (>>> 13.6 "Checking the safety configuration of the robot controller" Page 144)

> **⚠ WARNING** If the activation of a project fails, an error message is displayed in WorkVisual. In this case, one of the following measures must be carried out:
> - Either: Activate a project again (the same one or a different one).
> - Or: Reboot the robot controller with a cold restart.

4 Installation

4.1 PC system requirements

Hardware
Minimum requirements

- PC with Pentium IV processor, min. 1500 MHz
- 2 GB RAM
- 200 MB free hard disk space
- DirectX-compatible graphics card with a resolution of 1024x768 pixels

Recommended specifications

- PC with Pentium IV processor or higher and 2500 MHz
- 4 GB RAM
- 1 GB free hard disk space
- DirectX-compatible graphics card with a resolution of 1280x1024 pixels

Software
- Windows 7
  Both the 32-bit version and the 64-bit version can be used.

If Multiprog is to be interfaced with WorkVisual:

- KUKA.PLC Multiprog 5-35 4.1 must be installed.
- Multiprog must be licensed.

4.2 System requirements, robot controller

Software
- KUKA System Software 8.2, 8.3 or 8.4
- Or VW System Software 8.2 or 8.3

4.3 Installing WorkVisual

Precondition
- Local administrator rights

Procedure
1. Start the program **setup.exe**.
2. If the following components are not yet installed on the PC, an installation wizard opens:
   - .NET Framework 2.0, 3.0 and 3.5

   Follow the instructions in the installation wizard. .NET Framework is installed.
3. If the following component is not yet installed on the PC, an installation wizard opens:
   - SQL Server Compact 3.5

   Follow the instructions in the installation wizard. SQL Server Compact 3.5 is installed.
4. If the following components are not yet installed on the PC, an installation wizard opens:
   - Visual C++ Runtime Libraries
   - WinPcap

   Follow the instructions in the installation wizard. Visual C++ Runtime Libraries and/or WinPcap is installed.
5. The **WorkVisual […] Setup** window opens. Click on **Next**.
6. Accept the license agreement and click on **Next**.
7. Click on the desired installation type.

> ℹ️ Only carry out step 8 if the type **Custom** has been selected. Otherwise, continue with step 9.

8. The **Custom Setup** window opens.
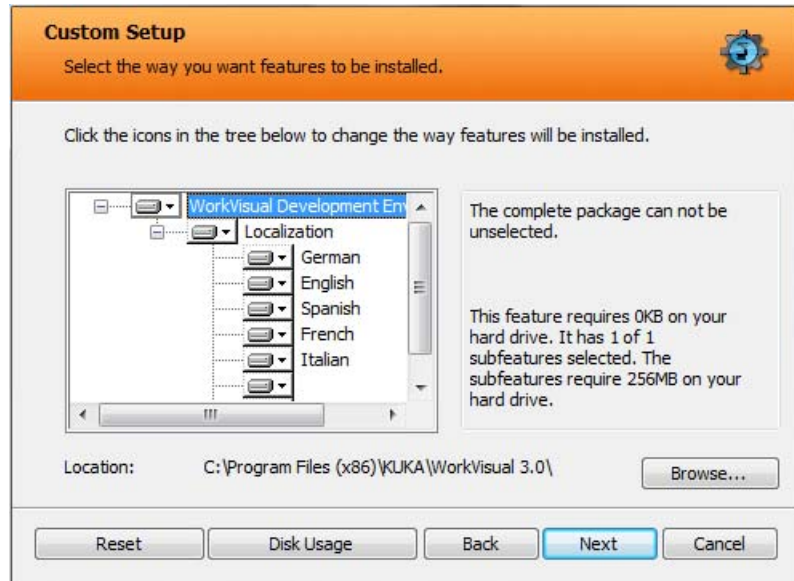
Fig. 4-1: Custom Setup window

a. If required, select a different installation directory with **Browse...**. If **Browse…** is grayed out: select the level **WorkVisual Development Environment** in the tree. **Browse…** is then enabled.

b. Select the desired languages in the tree. Only those languages that are installed here are available later for switching languages on the user interface.

Icon	Description
🗔	Will be installed.
🗔🔡	Will be installed, including subelements (not relevant for language selection)
✖	Will not be installed.

c. Continue with **Next**.

9. Click on **Install**. WorkVisual is installed.

10. Once installation is completed, click on **Finish** to close the installation wizard.

Installation types

Type	Installation directory	Languages
Typical	Default directory	English and the language of the operating system are installed.
Custom	Selectable	Selectable from list
Complete	Default directory	All are installed

Only those languages that are installed are available later for switching languages on the user interface.

Default directory: C:\Program Files (x86)\KUKA\WorkVisual […]

4.4 Uninstalling WorkVisual

> **i** It is advisable to archive all relevant data before uninstalling a software package.

Precondition
- Local administrator rights
- The setting **Show hidden files, folders and drives** is activated in Windows Explorer.

Procedure
1. In the Windows Start menu, delete the entry **WorkVisual […]** under **Control Panel** > **Remove Program**.

> **i** The remaining steps are only necessary if the entire user configuration is to be deleted.

2. Delete the **WorkVisual Projects** folder in the directory C:\User*Username*\My Documents.
3. In the directory C:\ProgramData\KUKA Roboter GmbH, delete the folders **DeviceDescriptions** and **WorkVisual**.

5 Graphical user interface

5.1 Overview of the graphical user interface

Not all elements on the graphical user interface are visible by default, but they can be shown or hidden as required.

There are other windows and editors available in addition to those shown here. These can be displayed via the **Window** and **Editors** menus.
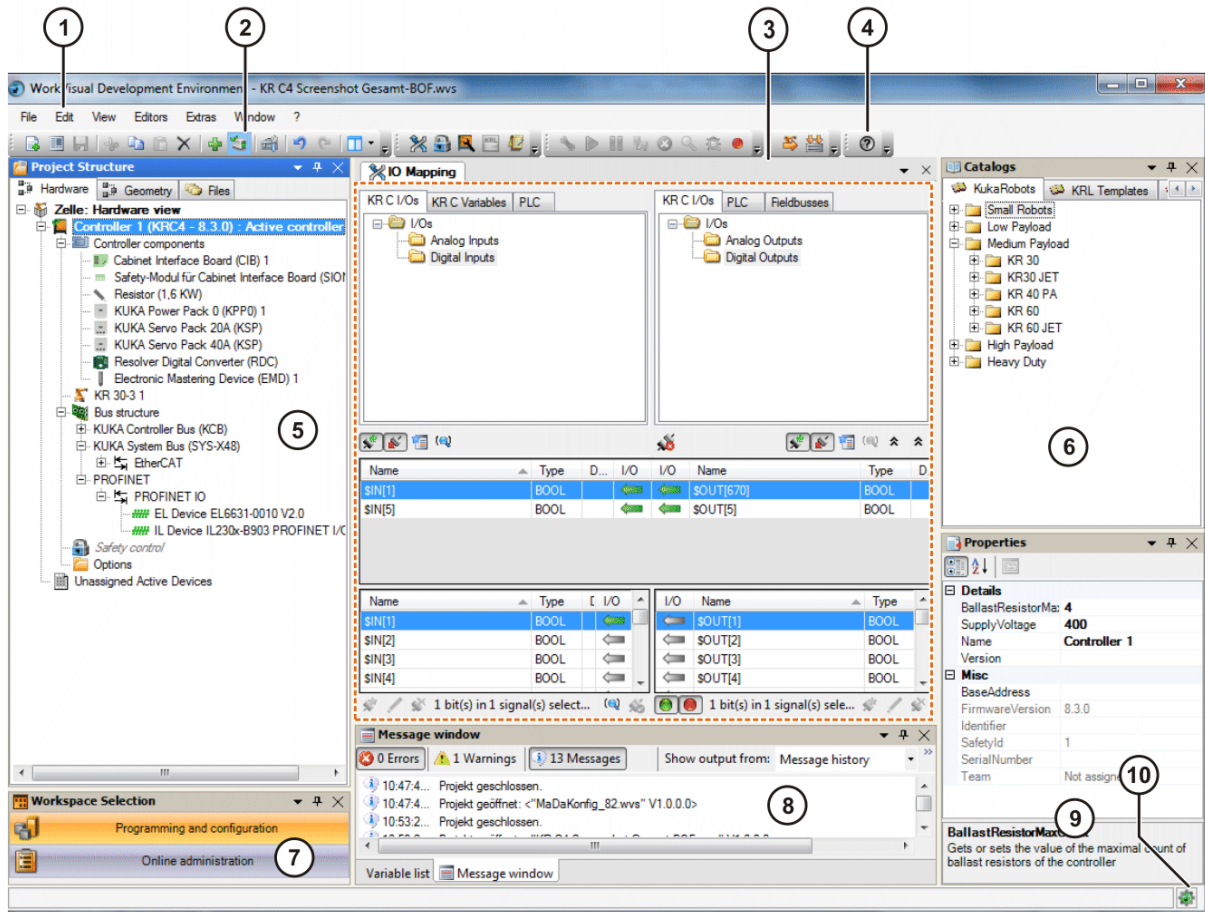
Fig. 5-1: Overview of the graphical user interface

Item	Description
1	Menu bar
2	Button bars
	(>>> 5.5 "Displaying/hiding buttons" Page 23)
	(>>> 5.6 "Button bar" Page 23)
3	Editor area
	If an editor is open, it is displayed here. More than one editor can be open simultaneously – as shown here in the example. In this case, they are stacked one on top of the other and can be selected via tabs.
4	Help button
5	**Project structure** window
	(>>> 5.8 "Project structure window" Page 26)

Item	Description
6	**Catalogs** window
	All inserted catalogs are displayed in this window. The elements in the catalogs can be inserted on the **Hardware** or **Geometry** tabs in the **Project structure** window.
7	**Workspaces** window
	(>>> 5.4 "Displaying different views of the user interface" Page 22)
8	**Messages** window
	(>>> 5.7 "Messages window" Page 26)
9	**Properties** window
	If an object is selected, its properties are displayed in this window. The properties can be changed. Individual properties in gray boxes cannot be changed.
10	**WorkVisual project analysis** icon
	(>>> 14.1 "Project analysis" Page 149)

5.2 Displaying/hiding windows

Procedure

1. Select the menu item **Window**. A list of available windows opens.
2. Click on a window in the list in order to display or hide it on the graphical user interface.

5.3 Repositioning the windows

Precondition

- The desired window is displayed on the graphical user interface.

Procedure

Free positioning of windows:

1. Right-click in the title bar of the window. A context menu is opened.
2. Select the option **Floating**.
3. Grip the window by the title bar and move it to the desired position on the graphical user interface.

If the mouse pointer is positioned over the edges or corners of the window, arrows appear which can be dragged to make the window larger or smaller.

Fixed positioning of windows:

1. Right-click in the title bar of the window. A context menu is opened.
2. Select the option **Dockable**.
3. Grip the window by the title bar and move it to the desired position on the graphical user interface.
   - Anchor points are displayed at the right, left, top and bottom of the user interface.
   - If the window is moved over another fixed window, an anchor cross is displayed.
     (>>> "Anchor cross" Page 21)
4. Drag the window onto an anchor point or the cross. The window is now anchored.

Automatic hiding and displaying of anchored windows:

1. Right-click in the title bar of the window. A context menu is opened.
2. Select the option **Auto-Hide**. The window is hidden. A tab with the name of the window remains at the edge of the user interface.

3. To show the window again, move the mouse pointer over the tab.

4. To hide the window again, move the mouse pointer off the window. If necessary, click on an area outside the window.

The **Auto-Hide** option creates more space for working in other areas of the user interface. Nonetheless, the window can still be quickly shown again at any time.

There is a pin symbol in the title bar of the window.

■ Alternatively, **Auto-Hide** can also be activated or deactivated by clicking on this pin symbol.
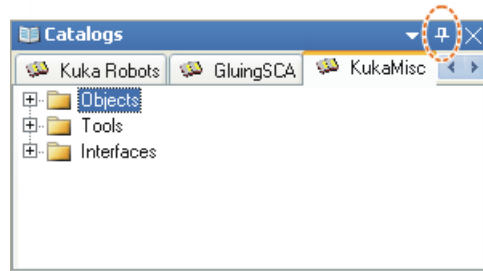
Fig. 5-2: Pin symbol

Anchor cross   If a window is moved over another fixed window, an anchor cross is displayed.
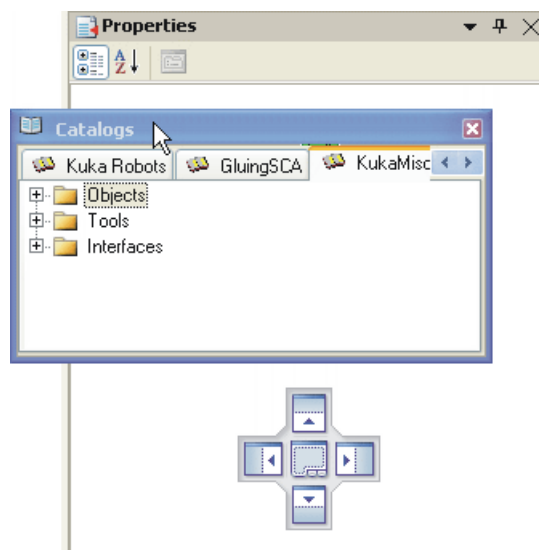
Fig. 5-3: Anchor cross

Depending on which side of the anchor cross the window is dragged, it is then anchored on this side of the previously window.

If the window is dragged over to the center of the anchor cross, both windows are anchored one on top of the other. Tabs are displayed underneath the windows, making it possible to switch between windows.
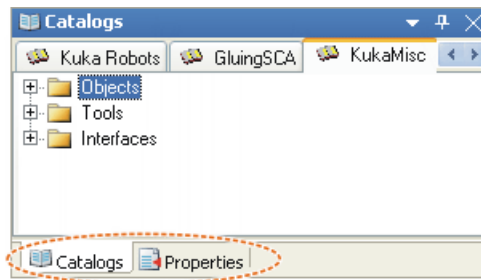
Fig. 5-4: Windows anchored on top of each other

To move windows that are anchored one on top of the other:

- Dragging on a tab moves only the corresponding window.
- Dragging on the title bar moves the whole stack of windows.

5.4 Displaying different views of the user interface

Description

The WorkVisual user interface can be displayed in 2 different views. These can be selected via the menu item **View** or via the **Workspaces** window.

The views are tailored to different types of work:

View	Type of work
Configuration and start-up	Area for project-related work e.g. cell configuration, I/O mapping and work with the KRL Editor.
Programming and diagnosis	Area for project-independent work e.g. monitoring, recording and work with the KRC Explorer. The functions in this view are only available when there is no project opened.

Each view can be adapted separately to the needs of the user. Examples:

- Position the button bars differently in each view.
- Hide the message window in one view and not in another.

Procedure

To display the **Workspaces** window:

- Select the menu sequence **Window** > **Workspaces**.
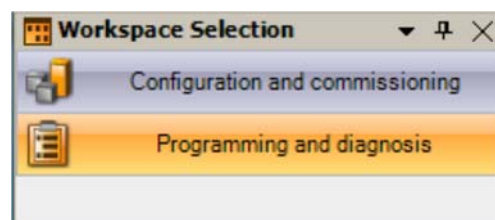
Fig. 5-5: "Workspaces" window

To reset the current view to the default settings:

- Select the menu sequence **Window** > **Reset active workspace**.

To reset all views to the default settings:

- Select the menu sequence **Window** > **Reset all workspaces**.

5.5 Displaying/hiding buttons

Description The individual buttons can be added or removed. In this way, the button bar can be tailored to the needs of the user.

Procedure 1. Click on the arrow on the right of the button bar.

Fig. 5-6: Example: File button bar: Click on the arrow on the right

2. The menu item **Add or Remove Buttons** is displayed. Click on this, and then on the submenu item **[***BarName***]**.

3. An overview of all the buttons in this bar is displayed. Click on a button in the overview in order to add or remove it.

	New...	Ctrl+N
	Open Project	Ctrl+O
	Save	Ctrl+S
	Cut	
	Copy	
	Paste	
	Delete	
	Add...	Ctrl+I
	Set as active controller	Ctrl+Shift+A
	Undo	
	Redo	
	Windows	
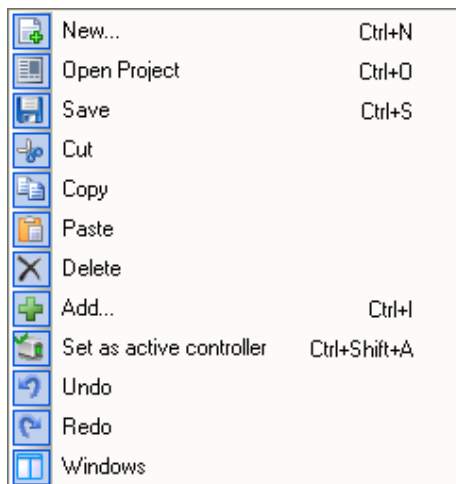
Fig. 5-7: Example: File button bar: Overview

5.6 Button bar

Button	Name / description
	New… Opens a new, empty project.
	Open project Opens the **Project Explorer**.
	Save project Saves the project.
	Cut Deletes the selected element from its original position and copies it to the clipboard.
	Copy Copies the selected element to the clipboard.
	Paste Inserts the cut or copied element at the selected position.
	Delete Deletes the selected element.

Button	Name / description
	Opens the Add Point dialog.
	Opens a window in which an element can be selected and added to the tree structure. Which element is available depends on what is selected in the tree structure.
	The button is only active if an element is selected on the **Project structure** or **Hardware** tab in the **Files** window.
	Set as active controller / **Reset active controller**
	Sets a robot controller to active/inactive.
	The button is only active if a robot controller is selected in the **Project structure** window.
	Configuration proposal...
	Opens a window in which WorkVisual proposes complete hardware configurations that match the existing kinematic systems. The user can select which proposal matches the real configuration and apply this configuration to the project
	Undo
	Undoes the last action.
	Restore
	Redoes the action that was undone.
	Settings…
	Opens a window with the hardware data.
	The button is only active if a device is selected on the **Hardware** tab in the **Project structure** window.
	Connect with device
	Establishes a connection with a field bus device.
	The button is only active if the field bus master is selected on the **Hardware** tab in the **Project structure** window.
	Disconnect from device
	Terminates the connection with a field bus device.
	Scan Topology…
	Scans a bus.
	Abort last action
	Cancels certain actions, e.g. a bus scan.
	The button is only active if a cancelable action is running.
	Monitor
	Currently without function.
	Diagnosis…
	Currently without function.
	Perform network capture...
	WorkVisual can record communication data from interfaces of the robot controller. This button opens the corresponding window.
	Deploy…
	Transfers the project to the robot controller.

Button	Name / description
	Generate code (>>> 13.1 "Generating code" Page 137)
	Tool/base management Opens a graphic editor for managing the TOOL and BASE co-ordinate systems. (>>> 6.24.1 "Opening tool/base management" Page 47)
	Mapping editor Opens the **I/O Mapping** window.
	The local safety configuration of the controller Opens the local safety configuration of the current robot controller.
	Drive configuration Opens a graphic editor for adapting the drive channels.
	KRL Editor Opens the selected file in the KRL Editor. The button is only active if a file is selected on the **Files** tab in the **Project structure** window which can be opened with the KRL Editor.
	Symbol Table Editor Opens the **Symbol Table Editor** window.
	Cell configuration Opens the **Cell configuration** window.
	Help Opens the Help.

Only in the **Programming and diagnosis** workspace:

Button	Name / description
	Opens the **Online system information** window. (>>> 14.6 "Displaying online system information" Page 170)
	Opens the **Diagnostic monitor** window. (>>> 14.5 "Displaying diagnostic data about the robot controller" Page 166)
	Opens the **Trace configuration** window. (>>> 14.2.4 ""Trace configuration" window" Page 152)
	Opens the **Trace Analysis (Oscilloscope)** window. (>>> 14.2.7 ""Trace Analysis" window" Page 158)
	Opens the **Cell definition** window. (>>> 6.25.1 "Opening the cell definition" Page 53)
	Opens the **Log view** window. (>>> 14.4 "Displaying messages and system logs of the robot controller" Page 163)
	Establishes a connection to the working directory of the real robot controller. (>>> 12.6.1 "Opening the working directory in the KRC Explorer" Page 126)

Button	Name / description
	Restores the state of the working directory of the real robot controller. (>>> 12.6.8 "Restoring the state of the robot controller" Page 130)
	Transfers the changes in the working directory from WorkVisual to the real robot controller. (>>> 12.6.6 "Transferring changes to the robot controller" Page 129)
	Loads changes from the robot controller. (>>> 12.6.7 "Loading changes from the robot controller" Page 130)
▶	Starts the debugging mode. (>>> 12.7.2 "Starting the debugging mode" Page 133)
■	Ends the debugging mode. (>>> 12.7.4 "Ending the debugging mode" Page 135)

The following buttons are only available when an OPS is used:

Button	Description
▶	Starts the program.
■	Stops the program.
↑	Resets the program.

(>>> 12.7.1 "Overview of the debugging mode" Page 132)

5.7 Messages window

Description Messages are displayed here. The following options can be set in the message window:

Language:

The desired language can be selected here.

Category:

- **Message history**: Displays all messages except errors relating to KRL code.

  Messages are not automatically deleted, even if they refer to a temporary state that is no longer active. Messages can be deleted by right-clicking on them and selecting the option **Delete all**.
- **KRL Parser**: Display of errors in the KRL code of the file currently open in the **KRL Editor** window.

5.8 Project structure window

The **Project structure** window contains the following tabs:

Hardware The **Hardware** tab shows the relationship between the various devices. Here, the individual devices can be assigned to a robot controller.

Geometry The **Geometry** tab displays all the geometrical objects used in the project in a tree structure (kinematic systems, tools, base objects). The properties of the objects can be edited.

If objects need to be linked geometrically, e.g. if a robot is to be assigned to a KUKA linear unit: this must be done here on the **Geometry** tab (Drag&Drop).

Files The **Files** tab contains the program and configuration files belonging to the project.

Coloring of file names:

- Files generated automatically (with **Generate code** function): gray
- Files inserted manually in WorkVisual: blue
- Files transferred to WorkVisual from the robot controller: black

5.9 Cell configuration window

Call ■ Select the menu sequence **Editors** > **Cell configuration**.

Description The **Cell configuration** window displays graphically certain geometric objects that are present in the project (robot controllers and kinematic systems). The objects can be linked together, e.g. a robot can be assigned to a robot controller. This is done by clicking on one of the objects and holding down the mouse button. The mouse pointer is then dragged to the other object and released there.

If a robot is linked to a different kinematic system, the type of link to be made can be selected via a pop-up window, e.g. whether the robot is to stand on the flange of the other kinematic system (geometric relationship) or whether the robot is to be able to track the other kinematic system (kinematic relationship).
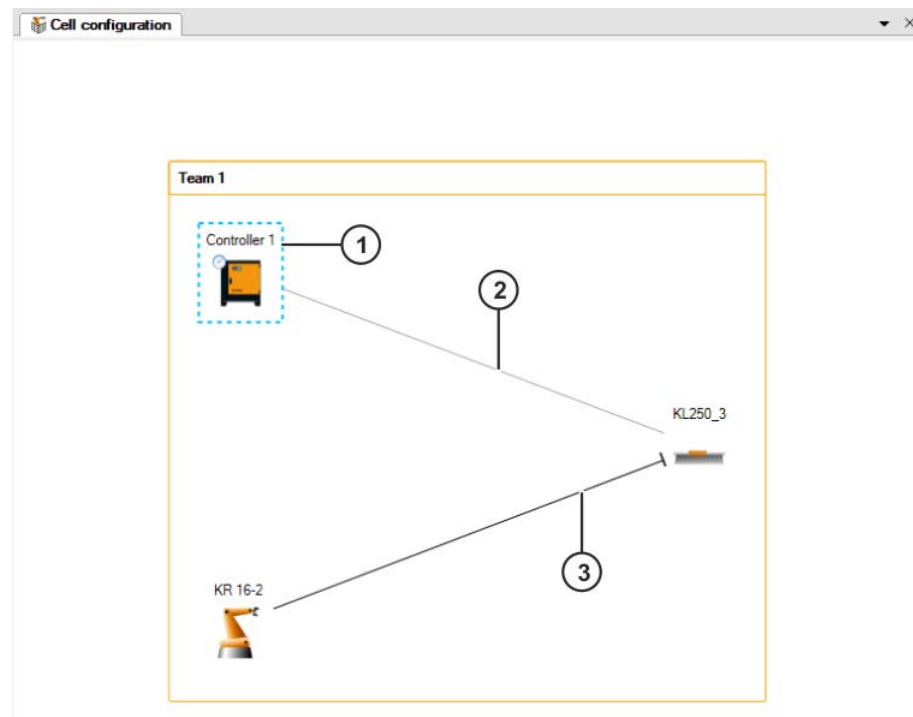
Fig. 5-8: Objects in the Cell configuration window

Item	Description
1	The dotted blue frame indicates that the object is selected.
2	The connecting line indicates that the KL is assigned to the robot controller.
3	The connecting line indicates that the robot is mounted on the KL.

5.9.1 Repositioning objects

Procedure
1. Move the mouse pointer onto the text over the object.
2. When the mouse pointer turns into a cross, click on it and drag the object to the desired position.

5.10 Resetting the graphical user interface

Description

All the settings affecting the graphical user interface and its behavior that the user has made in WorkVisual can be reset to the default state (as after installation).

This includes changes to the button bars, windows that have been displayed or hidden, and settings made in the **Options** window.

Procedure
1. Select the menu sequence **Window** > **Reset configuration**.
2. Close WorkVisual and restart it.

6 Operation

6.1 Starting WorkVisual

Procedure
1. Double-click on the WorkVisual icon on the desktop.
2. When WorkVisual is started for the first time, the DTM Catalog Management is opened. Here a catalog scan must be carried out.

   (>>> 6.9.1 "Updating the DtmCatalog (Catalog Scan)" Page 32)

6.2 Opening a project

Description
This procedure is used to open a project.

Projects can also be opened from older versions of WorkVisual. WorkVisual creates a backup copy of the older project and then converts the project. Before this happens, a dialog is displayed requesting the user to confirm the conversion.

Procedure
1. Select the menu sequence **File** > **Open project**.

   Or: Click on the **Open project** button.
2. The **Project Explorer** is opened. On the left, the **Open project** tab is selected. A list of projects is displayed.

   Select a project and click on **Open**. The project is opened.
3. Set the robot controller as the active controller.
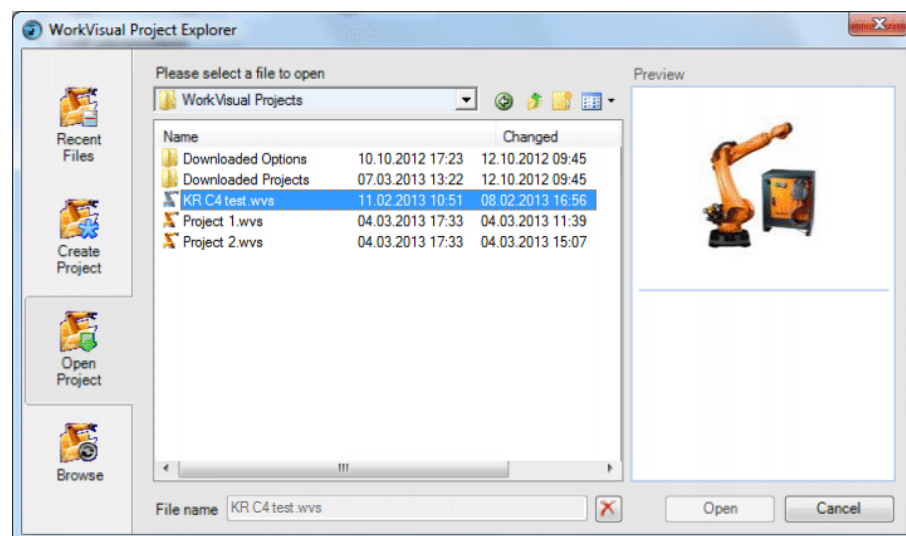
Fig. 6-1: Project Explorer

Alternative procedure
1. Select the menu sequence **File** > **Recent projects**. A submenu containing recently opened projects is opened.
2. Select a project. The project is opened.
3. Set the robot controller as the active controller.

Projects which are located on a robot controller and have not yet been saved on this PC can be loaded in WorkVisual and opened there. To do this, a different procedure must be used:

(>>> 13.7 "Loading the project from the robot controller" Page 144)

6.3 Creating a new project

6.3.1 Creating a new empty project

Procedure
1. Click on the **New…** button. The **Project Explorer** is opened. On the left, the **Create project** tab is selected.
2. Select the **Blank project** template.
3. Enter a name for the project in the **File name** box.
4. The default directory for projects is given in the **Location** box. If required, select a different directory.
5. Click on the **New** button. A new, empty project is opened.

6.3.2 Creating a project with a template

Procedure
1. Click on the **New…** button. The **Project Explorer** is opened. On the left, the **Create project** tab is selected.
2. Select the desired template in the **Available templates** area of the tab.
3. Enter a name for the project in the **File name** box.
4. The default directory for projects is given in the **Location** box. If required, select a different directory.
5. Click on the **New** button. The new project is opened.

Templates
The templates available for selection include:

Template	Description
Blank project	Blank project
KR C4 project	This project contains a KR C4 controller and the catalog **KRL Templates**.
VKR C4 project	This project contains a VKR C4 controller and the catalog **VW Templates**.

For templates with controller:

If the real robot controller is a VKR C4 retrofit, or if certain options are used, this must be activated in WorkVisual.

6.3.3 Creating a project on the basis of an existing project

Procedure
1. Click on the **New…** button. The **Project Explorer** is opened. On the left, the **Create project** tab is selected.
2. Select the desired project in the **Available projects** area of the tab.
3. Enter a name for the new project in the **File name** box.
4. The default directory for projects is given in the **Location** box. If required, select a different directory.
5. Click on the **New** button. The new project is opened.

6.4 Saving project information

Description
If required, the user can assign information to the project. The following information can be saved:

- A description
- A preview image (screenshot)

The following information is saved and displayed by default:

- The version
- The creation date and change date
- The current file location
- The project ID

Precondition

- A project is open.

Procedure

1. Select the menu sequence **Extras** > **Project info**. The window **Project info for ...** is opened.
2. Enter a description in the **Description** box (optional).
3. Click on the **Insert image from file** button (optional). Select an image and confirm with **Open**.
4. Click on **OK**. The window **Project info for ...** is closed and the project information is saved.

6.5 Saving the project

Description

Projects have the file format WVS ("WorkVisual Solution").

A project can be saved with one of the following functions:

- **Save**: Saves the open project.
- **Save as**: This function is used to save a copy of the open project.

  The open project itself is closed and remains unchanged.

Procedure for Save

- Select the menu sequence **File** > **Save**.

  Or click on the **Save project** button.

Procedure for Save as

1. Select the menu sequence **File** > **Save as**.

   The **Save as** window is opened. A file location for the project can be selected here.
2. Enter a name in the **File name** box and click on the **Save** button.

6.6 Closing the project

Procedure

- Select the menu sequence **File** > **Close**.

  If changes have been made, a request for confirmation is displayed, asking if the project should be saved.

6.7 Closing WorkVisual

Procedure

- Select the menu sequence **File** > **Exit**.

  If a project is open, a request for confirmation is displayed, asking if the project should be saved.

6.8 Importing device description files

Description

To be able to use a device in WorkVisual, WorkVisual requires the device description file for this device.
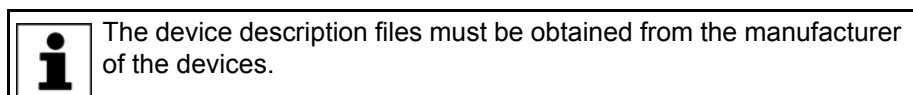
> **i** The device description files must be obtained from the manufacturer of the devices.

Precondition

- There is no project open.

Procedure	1. Select the menu sequence **File** > **Import / Export**. A window opens.
	2. Select **Import device description file.** and click on **Next >**.
	3. Click on **Browse…** and navigate to the directory where the files are located. Confirm with **Next >**.
	4. Another window opens. Select the required type in the **File type** box. For devices for KUKA buses, the type **EtherCAT ESI** must be selected.
	5. Select the file to be imported and confirm with **Open**.
	6. Click on **Finish**.
	7. Close the window.
	The imported file is now available in the **DtmCatalog**.

6.9 Catalogs

6.9.1 Updating the DtmCatalog (Catalog Scan)

Description	This procedure is used to update the **DtmCatalog**. As a rule, it only needs to be carried out when WorkVisual is started for the first time after installation or after an update.
	An exception is EtherNet/IP if an EDS file was imported. After this, the catalog scan must be carried out.
Precondition	■ There is no project open.
Procedure	1. The **DTM Catalog Management** window opens automatically. If necessary, it can also be opened via the menu sequence **Extras** > **DTM Catalog Management...**.
	2. Click on **Search for installed DTMs**. WorkVisual searches the PC for relevant files. The search results are displayed.
	3. Under **Known DTMs**, select the required files and click on the **Right arrow** button. If all files are to be accepted, click on the **Double right arrow** button.
	4. The selected files are displayed under **Current DTM Catalog**. Click on **OK**.
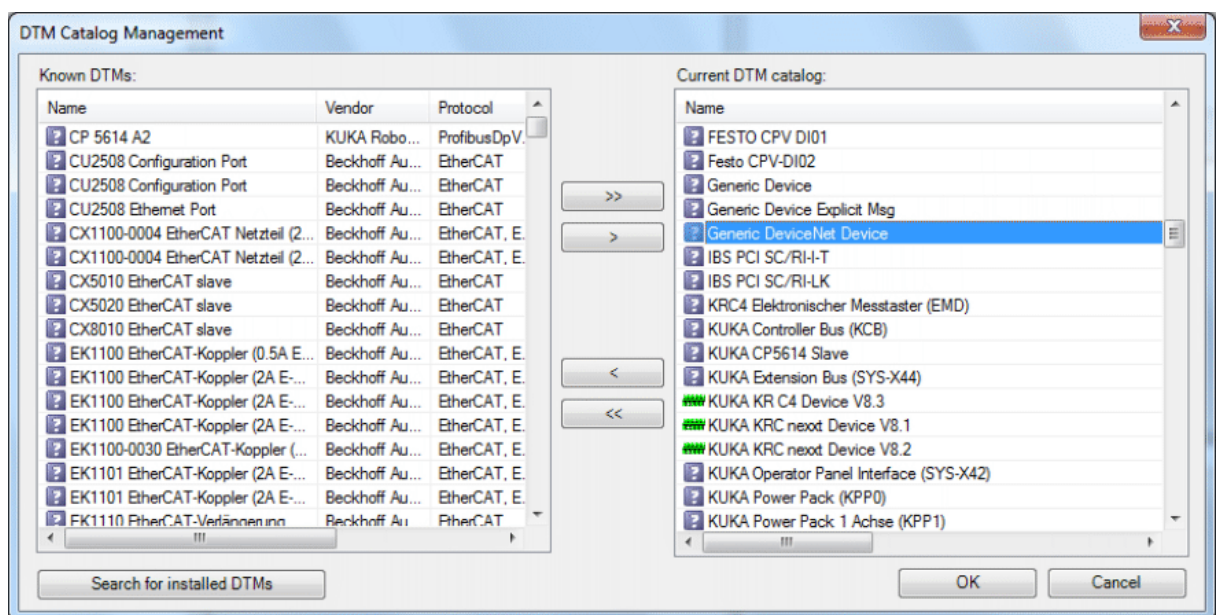
Fig. 6-2: DTM Catalog Management

6.9.2 Inserting a catalog in a project

Description The catalogs contain all the elements that are required to generate programs. In order to be able to use a catalog, it must first be inserted in the project.

Procedure
1. Select the menu sequence **File** > **Catalog management...**. A window opens.
2. Double-click on the desired catalog in the **Available catalogs** area. The catalog is now displayed in the **Project catalogs** area.
3. Close the window.

   The catalog has been inserted in the project. It is now available in the **Catalogs** window.

6.9.3 Adding a catalog

Description Catalogs created using the WorkVisual catalog editor can be added to the catalogs in WorkVisual.

Procedure
1. Select the menu sequence **File** > **Catalog management...**. A window opens.
2. Click on the [ ] button in the bottom left-hand corner. A window opens.
3. Select the desired catalog and click on **Open**.
4. Close the window. The catalog is now available in the **Catalogs** window.

6.9.4 Removing a catalog from the project

Procedure
1. Select the menu sequence **File** > **Catalog management...**. A window opens.
2. Double-click on the catalog to be removed in the **Project catalogs** area. The catalog is now displayed in the **Available catalogs** area.
3. Close the window.

6.9.5 Catalog descriptions (8.2)

Which catalogs are available depends on whether a robot controller with version 8.2 or 8.3 is used.

Catalog	Catalog contains ...
DtmCatalog	Device description files **Note:** The robot controller must already have been set once as the active controller in order for this catalog to be used.
KRL templates	KRL program templates
KukaControllers	Robot controllers, hardware components for robot controllers, safety options, PROCONOS option
KukaExternalAxes	KUKA linear units, KUKA turn-tilt table **External Kinematics Templates**: Templates for external kinematic systems that are not supplied by KUKA. These templates can be used if the machine data belonging to the kinematic system are only available in the form of XML files.

Catalog	Catalog contains ...
KUKARobots[…]	KUKA robots
KukaSpecialRobots	KUKA robots for special applications ■ **Food**, **Foundry**, **Clean Room**
MGU_Motor-Gear-Unit	KUKA motor/gear units An element from this catalog is used if a non-KUKA external axis fitted with a KUKA motor/gear unit is used with the real controller.
Motor_als_Kinematik	KUKA motors An element from this catalog is used if a non-KUKA external axis fitted with a KUKA motor is used with the real controller.
VW Templates	VW program templates

6.9.6 Catalog descriptions (8.3)

Which catalogs are available depends on whether a robot controller with version 8.2 or 8.3 is used.

Catalog	Catalog contains ...
Options	All options that are installed in WorkVisual
DtmCatalog	Device description files **Note:** The robot controller must already have been set once as the active controller in order for this catalog to be used.
KRL Templates	KRL program templates
KukaControllers	Robot controllers, hardware components for robot controllers, safety options, PROCONOS option
KukaExternal-Kinematics[…]	KUKA linear units, KUKA positioners
KukaRobots[…]	KUKA robots
VW Templates	VW program templates

6.10 Inserting an element in the project

Description

The procedure for inserting an element, e.g. a robot controller, is described separately for each individual element in this documentation. The Drag&Drop procedure is described.

In addition to Drag&Drop, the elements can usually also be inserted via the context menu. This procedure is not additionally mentioned in the individual descriptions, but is described here in general terms.

An element can be inserted into a project via the **Project structure** or **Cell configuration** window.

Procedure: Project structure

1. Right-click on the node in the tree structure under which the element is to be inserted. The type of node depends on the specific element concerned. The context menu is opened.
2. Select the **Add…** option from the context menu. A window opens.
3. Select the desired element in the window and accept it with **Add** or **OK**.

Procedure: Cell configuration

1. Right-click in the empty area in the **Cell configuration** window. The context menu is opened.
2. Select the **Add…** option from the context menu. A window with catalogs opens.
3. Select the catalog in which the desired element file is located.
4. Select the element and confirm with **Add**.

6.11 Removing an element from the project

Description

Inserted elements can be removed again from all tabs in the **Project structure** window. It is also possible to remove some – but not all – of the default elements there. The tree structure itself cannot be removed.

All elements can be removed again from the **Cell configuration** window.

Procedure

■ Right-click on the element. Select **Delete** from the context menu.
 Or:
■ Select the element. Select the menu sequence **Edit** > **Delete**.
 Or:
■ Select the element. Click on the **Delete** button in the toolbar, or press the Delete key on the keyboard.

6.12 Inserting a robot controller

Description

One or more robot controllers can be inserted in a project.

Precondition

■ The catalog **KukaControllers** is available.

Procedure: Project structure

1. Select the **Hardware** tab in the **Project structure** window.
2. Select the required robot controller in the **KukaControllers** catalog.
3. Drag the robot controller to **Cell: Hardware view** on the **Hardware** tab.

Procedure: Cell configuration

1. Select the required robot controller in the **KukaControllers** catalog.
2. Drag the robot controller into the **Cell configuration** window by means of Drag&Drop.

If the real robot controller is a VKR C4 retrofit, or if certain options are used, this must be activated in WorkVisual.

(>>> 6.16 "Activating additional controller settings" Page 37)

6.13 Setting the robot controller to active/inactive

Description

Most settings, actions and configurations performed in the **Configuration and start-up** workspace are only possible if a robot controller is active. Furthermore, they apply to the currently active robot controller (e.g. settings in the safety configuration and I/O mappings).

	If a project contains more than one robot controller, ensure that the correct robot controller is active.

Precondition

■ A robot controller has been added.

Procedure

Setting the robot controller as the active controller:

1. Double-click on the inactive robot controller on the **Hardware** tab in the **Project structure** window.

Or: Double-click on the inactive robot controller in the **Cell configuration** window.

2. Only if the robot controller is set to active for the first time: A window opens.

- **Firmware version** box: Enter the version of the KUKA/VW System Software that is installed on the real robot controller: e.g. "8.2.15".
- **Number of I/Os** box: Select the maximum number of inputs/outputs used on the robot controller.

The values can be changed subsequently. The correct values are necessary for code generation and project deployment.

(>>> 6.14 "Changing the values "Firmware version" and/or "Number of I/Os" Page 36)

3. Click on **OK** to save.

Instead of double-clicking, it is also possible to right-click on the robot controller. A context menu is opened. Select the option **Set as active controller**.

Setting the robot controller to inactive:

For a small number of actions in WorkVisual, it is necessary to set the robot controller to inactive. If these actions are started, a message is displayed to inform the user that the robot controller must first be set to inactive.

1. Save the project.
2. Double-click on the active robot controller on the **Hardware** tab in the **Project structure** window.

Or: Double-click on the active robot controller in the **Cell configuration** window.

Instead of double-clicking, it is also possible to right-click on the robot controller. A context menu is opened. Select the option **Reset active controller**.

6.14 Changing the values "Firmware version" and/or "Number of I/Os"

Description   When the robot controller is set to active for the first time, the values **Firmware version** and **Number of I/Os** must be adapted or confirmed.

(>>> 6.13 "Setting the robot controller to active/inactive" Page 35)

The values can be changed subsequently. The correct values are necessary for code generation and project deployment.

Procedure
1. Save the project.
2. Right-click on the robot controller on the **Hardware** tab in the **Project structure** window.

Or: Right-click on the robot controller in the **Cell configuration** window.

3. Select **Controller options** from the context menu. The **Controller options** window opens.
4. Enter the new value in the **Firmware version** box: e.g. "8.2.16".

And/or: In the **Number of I/Os** box, select a different number.

5. Click on **OK** to save.

6.15 Assigning a robot to the robot controller

Precondition
- The catalog containing the required robot has been inserted in the **Catalogs** window.
- The robot controller has been set as the active controller.

Procedure: Project structure
1. Select the **Hardware** tab in the **Project structure** window.

2. Select the required robot in the **KukaRobots[…]** catalog in the **Catalogs** window.

3. Drag the robot onto the robot controller in the **Hardware** tab. (Not onto the node **Unassigned active devices**.)

    The robot is now displayed underneath the robot controller.

Procedure: Cell configuration

1. Select the required robot in the **KukaRobots[…]** catalog in the **Catalogs** window.

2. Drag the robot onto the robot controller in the **Cell configuration** window.

    The robot is inserted and assigned to the robot controller.

6.16 Activating additional controller settings

Description

If one of the following options is used, it must be activated and set in WorkVisual:

- **Transformer ((V)KR C4)**
- **Air conditioner ((V)KR C4)**
- **Fast Measurement inputs ((V)KR C4)**
- **Retrofit interface E2/E7 (VKR C4)**
- **Braking mode settings**

  (>>> 6.16.1 "Braking mode settings" Page 37)

- **Transfer application**

  (>>> 6.16.2 "Setting the cycle time of the Transfer application" Page 38)

- **Force brake test**

  (>>> 6.16.3 "Force brake test" Page 39)

- **Tools and bases**

  (>>> 6.16.4 "Changing the number of TOOL and BASE coordinate systems" Page 39)

Precondition

- The project contains a robot controller.
- Only for **Fast Measurement inputs ((V)KR C4)**: A robot has been assigned to the robot controller.

Procedure

1. Select the menu sequence **Editors** > **Additional controller settings**. The **Additional controller settings** window opens.

2. Activate the check boxes for the options used or set the desired values.

3. Save the project.

6.16.1 Braking mode settings

Description

The behavior of the robot's axis brakes is defined in the **Braking mode settings**.

Option	Description
Brakes of robot axes are closed at the end of the command	■ **Activated**: The axis brakes of the robot are applied when the jog key is released. ■ **Deactivated**: The axis brakes of the robot are not applied when the jog key is released. Default setting: Activated
Brakes of all axes are opened/closed individually	■ **Activated**: The axis brakes of the robot are released and applied individually in axis-specific jogging. ■ **Deactivated**: The axis brakes of the robot are released and applied together. (All the brakes are connected to a single channel.) Default setting: Deactivated
Brakes close during motion pauses	■ **Activated**: The axis brakes of the robot are always applied together during motion pauses in programs. ■ **Deactivated**: The axis brakes of the robot are not applied during motion pauses in programs. Default setting: Activated
Brakes of external axes are closed/ opened independently of robot axes	■ **Activated**: Mathematically coupled external axes behave like the robot axes. External axes that are not mathematically coupled function independently of the robot axes if they are controlled separately. ■ **Deactivated**: The external axes behave like the robot axes according to the other braking mode settings. Default setting: Deactivated

6.16.2 Setting the cycle time of the Transfer application

Description

The **Transfer application** is used to transfer data from one field bus to another. The cycle time of this data transfer can be set manually or automatically.

> The larger the number of mappings and the shorter the cycle time, the higher the CPU load will be. The maximum number of possible mappings and the minimum possible cycle time have been chosen to ensure that the maximum possible CPU load resulting from the transfer application only places a minor load on the system. With certain system variants and configurations, however, it is nonetheless possible for the system to become overloaded. It is therefore recommended at start-up to use the diagnostic monitor to check the CPU load arising from the transfer application. (>>> 14.5.3 "Diagnostic data for the transfer application module" Page 169) If the CPU load is greater than 5%, it is recommended to activate automatic adaptation of the cycle time.
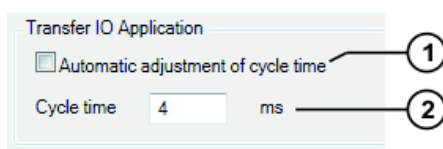
Fig. 6-3: Transfer application

Item	Description
1	**Automatic adaptation of cycle time** ■ **Activated**: The cycle time is automatically adjusted by the robot controller, but is not faster than 4 ms. ■ **Deactivated**: The cycle time can be entered manually in the **Cycle time** box.
2	Manual setting of the cycle time ■ **4 … 1000 ms**

6.16.3 Force brake test

Option	Description
Force brake test	■ **Activated**: The brake test is active. ■ **Deactivated**: The brake test is activated if a safety option is installed and the safe monitoring is activated. In all other cases, the brake test is deactivated.

6.16.4 Changing the number of TOOL and BASE coordinate systems

Option	Description
Number of tools	Number of TOOL coordinate systems ■ **16 … 128** Default value: 16
Number of bases	Number of BASE coordinate systems ■ **32 … 128** Default value: 32

6.17 Inserting a safety option and/or PROCONOS

Description

If a safety option (e.g. SafeOperation) and/or PROCONOS is used on the real robot controller, these options must also be inserted into the WorkVisual project.

Precondition

■ The catalog **KukaControllers** is available.

Procedure: Project structure

1. Select the **Hardware** tab in the **Project structure** window.
2. Expand the **Options** node in the **KukaControllers** catalog.
3. Drag the option onto the **Options** node on the **Hardware** tab.

When the option has been inserted, the version number is also displayed to the right of the name. It is always the compatible version for the robot controller.

Procedure: Cell configuration

1. Right-click on the robot controller in the **Cell configuration** window and select the option **Add…** from the context menu.
2. Select the option on the **KukaControllers** tab and accept it with **Add**. The option is inserted into the **Project structure** window.

6.18 Inserting hardware components

Description
The default hardware components for the robot controller are automatically present under the **Controller components** node.

If further components are present in the real robot controller, these must be added here. This can be done in one of the following ways:

- The components can be inserted individually.
- Or, for 8.3 only: WorkVisual can propose complete hardware configurations, one of which can be selected. The proposal always includes the hardware for all robots and external kinematic systems assigned to the robot controller.

Precondition
- The catalog **KukaControllers** is available.

Procedure
Inserting an individual component

1. Select the **Hardware** tab in the **Project structure** window.
2. Select the desired component in the **KukaControllers** catalog.
3. Drag the component onto the **Controller components** node on the **Hardware** tab.

For 8.3 only: Selecting a configuration proposal

1. Select the **Hardware** tab in the **Project structure** window.
2. Select the **Controller components** node and click on the **Configuration proposal...** button.

   The **Configuration proposal** window opens. The commonest configuration for this controller and the associated kinematic systems is displayed.
3. If this configuration corresponds to the real configuration, confirm with **Accept**. The configuration is inserted in the **Controller components** node.

   If not, expand the **Further proposals** area and click on the matching configuration. It is now displayed in the upper part of the window and can be accepted.

Configuration proposal
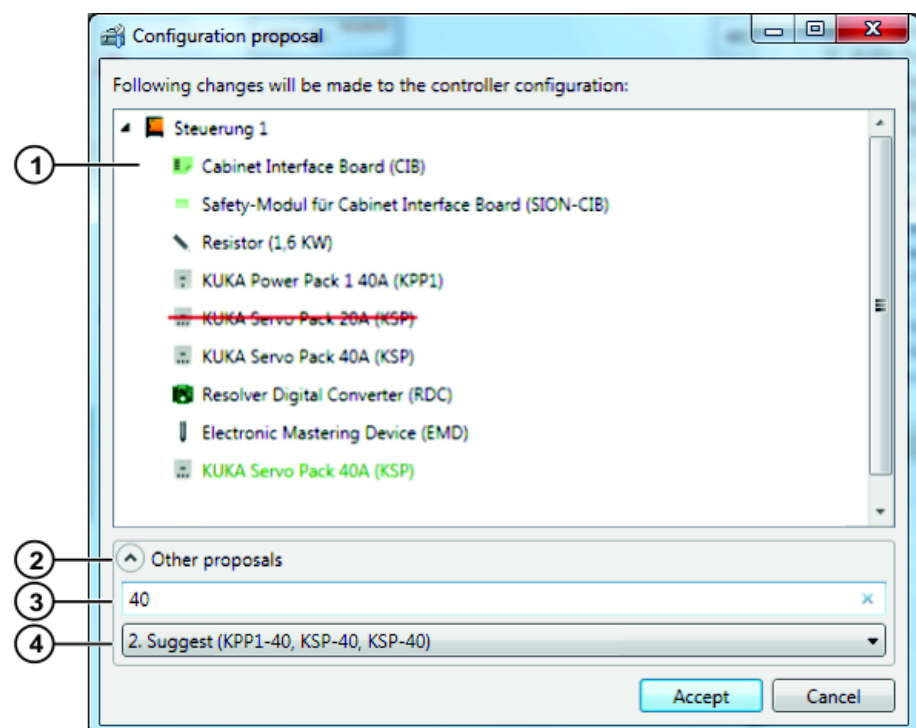
Fig. 6-4: "Configuration proposal" window

Item	Description
1	The selected proposal is displayed here.
	■ Black text: Component that is already present under **Controller components** and which would remain present if the proposal is accepted.
	■ Green text: Component which would be inserted.
	■ Crossed-out component: Component which would be removed.
2	Items 3 and 4 can be displayed or hidden by clicking on the arrow.
3	Here the proposals displayed under item 4 can be filtered. If no filter is entered, all possible configurations for this controller and the associated kinematic systems are displayed.
4	This box can be expanded to display the list of proposals. Click on a proposal to select it.

6.19 Inserting an external axis

Preparation

In order for an external axis to be inserted into the project, the file structure of the robot controller must be present on the **Files** tab in the **Project structure** window. This can be achieved as follows:

■ Do not create a new project in WorkVisual, but load the initial project from the robot controller.

(Via **File** > **Browse for project**.)

 (>>> 13.7 "Loading the project from the robot controller" Page 144)

■ Or: Transfer the project to the robot controller. Then transfer it back to WorkVisual via the menu sequence **Extras** > **Comparing projects**.

 (>>> 13.8 "Comparing projects (and accepting differences)" Page 145)

Precondition

■ The required catalog has been inserted in the **Catalogs** window.

■ The robot controller has been set as the active controller.

Procedure
Project structure

1. Select the **Hardware** tab in the **Project structure** window.

2. Select the external axis in the catalog in the **Catalogs** window.

3. Drag the external axis onto the robot controller in the **Hardware** tab. (Not onto the node **Unassigned active devices**.)

The external axis is now displayed underneath the robot controller.

4. Double-click on the external axis. The **Machine data configuration** editor is opened.

5. For 8.2 only:

In the **Axis ID** box in the area **General axis-specific machine data**, specify which drive the external axis is assigned to in the real cell.

6. If required: edit the other parameters.

 (>>> "Editor (8.2)" Page 43)

 (>>> "Editor (8.3)" Page 44)

7. If the external axis needs to be linked geometrically to a kinematic system:

a. Select the **Geometry** tab.

b. Assign the kinematic systems to each other as required by means of Drag&Drop.

Example 1: If a KUKA linear unit has been added, drag the robot onto the linear unit.

Example 2: If a servo gun (KUKA.ServoGun package) has been added that is to be used on the robot flange, drag the gun onto the **Flange Base** node of the robot.

Procedure Cell configuration	1.	Select the catalog for the external axis in the **Catalogs** window.
	2.	Select the desired external axis.

3. Drag the external axis onto the robot controller in the **Cell configuration** window. The external axis is inserted and assigned to the robot controller.

4. Double-click on the external axis. The **Machine data configuration** editor is opened.

5. For 8.2 only:

   In the **Axis ID** box in the area **General axis-specific machine data**, specify which drive the external axis is assigned to in the real cell.

6. If required: edit the other parameters.

   (>>> "Editor (8.2)" Page 43)

   (>>> "Editor (8.3)" Page 44)

7. If the external axis needs to be linked to a kinematic system:

   a. Click on a kinematic system and hold down the mouse button.

   b. Drag the mouse pointer to the other kinematic system and release the mouse button. A pop-up window opens. The type of link is defined here:

   {Kinematic1} should follow {Kinematic2}: kinematic relationship

   Place {Kinematic1} on flange of {Kinematic2}: geometric relationship

   The geometry is adapted automatically.

6.20 Editing machine data for external axes (8.2)

> **i** Depending on the firmware version of the robot controller (8.2 or 8.3), a different editor opens for editing machine data. If the robot controller has not yet been assigned a firmware version, the editor for 8.3 opens.

Description   If a robot controller with version 8.2 is part of the project, only the machine data of external axes can be edited.

> **i** The machine data must be edited using the procedure described here, i.e. via the editor. Direct editing of files, e.g. $machine.DAT, is not permitted in WorkVisual. The files are overwritten with the data from the editor during code generation.

Precondition   ■ The robot controller has been set as the active controller.

Procedure
1. Double-click on any kinematic system on the **Hardware** tab in the **Project structure** window. The editor is opened.

   Or: Double-click on any kinematic system in the **Cell configuration** window. The editor is opened.

2. In the editor, select the kinematic system that is to be edited.

3. Edit the machine data as required.

4. Save the project to accept the changes.

KUKA

Editor (8.2)

Fig. 6-5: Machine data configuration editor (8.2)

Item	Description
1	Here, select the kinematic system that is to be edited.
2	The product name of the selected kinematic system is displayed here. The box cannot be edited.
3	With one exception, this button never needs to be pressed. If an import is necessary, WorkVisual performs this automatically at the relevant time. Exception: If an XML file with machine data for a non-KUKA kinematic system has been added on the **Files** tab using **Add external file**, then the machine data must subsequently be imported manually from the XML file in a second step. This is done using this button.
4	These boxes have no effect.
5	The machine data of the selected kinematic system are displayed here, sorted by axis.
6	The data belonging to each heading can be displayed or hidden by clicking on the arrow.
7	Motor data The data can be displayed by clicking on the box.

6.21 Editing machine data (8.3)

> **i** Depending on the firmware version of the robot controller (8.2 or 8.3), a different editor opens for editing machine data. If the robot controller has not yet been assigned a firmware version, the editor for 8.3 opens.

Description   Each kinematic system and each subordinate element belonging to it has its own editor with the respective data. A subordinate element can be an axis or a motor, for example. More than one editor can be open simultaneously.

> ℹ️ The machine data must be edited using the procedure described here, i.e. via the editor. Direct editing of files, e.g. $machine.DAT, is not permitted in WorkVisual. The files are overwritten with the data from the editor during code generation.

Procedure

1. On the **Hardware** tab in the **Project structure** window, double-click on the element to be edited. The editor is opened.

   Or: In the **Cell configuration** window, double-click on the element to be edited. The editor is opened.

2. Edit the machine data as required.

3. Save the project to accept the changes.

Editor (8.3)

> ℹ️ Information about the individual machine data can be found in the documentation **Configuration of Kinematic Systems**.
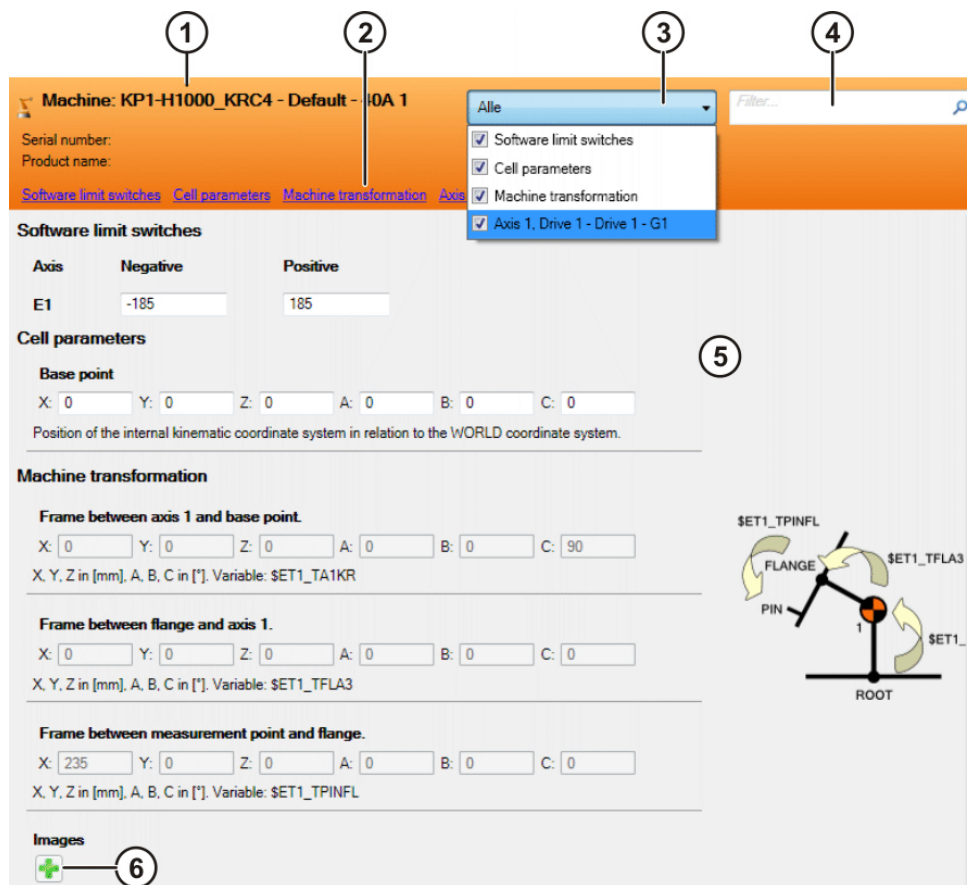
Fig. 6-6: Machine data configuration editor (8.3)

Item	Description
1	The name of the kinematic system is displayed here.
2	The parameter groups for this kinematic system are displayed here. Clicking on one group shows this group in the parameter display and hides all other groups.
3	This selection box indicates which groups are currently shown in the parameter display. The selection list contains all the existing groups. The groups can be displayed or hidden using the check boxes.

Item	Description
4	The parameter display can be filtered here. The filter refers to the parameter name. No distinction is made between uppercase and lowercase letters.
	Example: If *a1* is entered, only those parameters are displayed which have *a1* or *A1* in their names.
5	Parameter display (area with gray background)
	The parameters are sorted by group. They can be edited. If a parameter is changed, the value is displayed in blue until the change is saved. In addition, the tab of the editor is then marked with an asterisk (not shown) until the change is saved.
6	Graphics files can be loaded here. Once a file has been loaded, a minus symbol is also displayed, which can be used to delete it again.
	The graphic is shown here. Only one graphic is displayed at a time. If several files are loaded, a selection box is displayed for toggling between the graphics.
	Formats: JPG, JPEG, PNG, BMP

6.22   Comparing kinematic systems (and accepting differences)

Description   A kinematic system in WorkVisual can be compared with another kinematic system. This may be a kinematic system in a catalog or in a different project. The differences are clearly listed. The user can decide for each individual difference whether to retain the state of the current kinematic system or to adopt the state of the other kinematic system.

Precondition   ■   The kinematic system that is to be compared is contained in the open WorkVisual project.

Procedure   1.   Expand the tree structure of the robot controller on the **Hardware** tab in the **Project structure** window.

2.   Right-click on the kinematic system and select **Compare kinematic machines**. Then select the desired option:

   ■   **From open Catalog**: if the kinematic system to be compared is in an open catalog. The kinematic systems are compared and the differences are displayed in an overview (>>> Fig. 6-7). Continue with step 5.

   ■   **Select catalog**: if the kinematic system to be compared is in a catalog that is not open. Continue with step 3.

   ■   **Select project**: if the kinematic system to be compared is in a different project. Continue with step 3.

3.   A window opens. Select the catalog or project in which the kinematic system is located and click on **Open**.

4.   Open the tree structure, select the kinematic system and click on **Compare**. The differences between the kinematic systems are displayed in an overview (>>> Fig. 6-7).

5.   For each difference, select whether to retain the state of the current kinematic system or to adopt the state of the comparison kinematic system. This does not have to be done for all the differences at one go.

   If suitable, the default selection can also be accepted.

6.   Click on **Merge** to accept the changes.

7.   Repeat steps 5 to 6 as required. This makes it possible to work through the different areas bit by bit.

8.   Close the **Compare kinematic machines** window.

Comparison The differences between the kinematic systems are displayed in an overview. For each difference, the user can select which state to accept. The default setting is as follows:

- For all elements that are present in the kinematic system in the open project, the state of this kinematic system is selected.
- For all elements that are not present in the kinematic system in the open project, the state of the comparison kinematic system is selected.

Fig. 6-7: Example: overview of differences

Item	Description
1	The node for the kinematic system. The different areas are represented by subnodes. The nodes can be expanded to display the comparisons.
	■ In each line, place a check mark in the box for the value that should be transferred.
	■ A check mark next to **Not available** means that the element will not be transferred, or that it will be deleted from the kinematic system if already present.
	■ If a check box is activated for a node, the check boxes for all subordinate elements are also automatically activated.
	If a check box is deactivated for a node, the check boxes for all subordinated elements are also automatically deactivated.
	The subordinate elements can also be edited individually, however.
	■ A filled-in box means: at least one of the subordinate elements is selected, but not all.
2	State of the kinematic system in the project that is open in WorkVisual
3	State of the comparison kinematic system

Item	Description
4	Back arrow: The focus in the display jumps to the previous difference. Collapsed nodes are automatically expanded.
5	Forward arrow: The focus in the display jumps to the next difference. Collapsed nodes are automatically expanded.
6	**Activated**: Only the areas containing differences are displayed. All other areas are hidden.
7	Transfers the selected changes to the kinematic system in the open project.
8	Click on a marking. The focus of the display jumps to the corresponding difference.

6.23 Exporting kinematic systems to a catalog

Description   This procedure can be used to export kinematic systems to a catalog. In this way, the kinematic systems can be used in different projects.

Precondition   ■   A project is open.

Procedure
1. Right-click on (any) kinematic system on the **Hardware** tab in the **Project structure** window and select **Export kinematic system** from the context menu. A window opens.
2. All controllers and kinematic systems in the project are displayed in a tree structure. Activate the check boxes next to the kinematic systems that are to be exported. Confirm with **Next >**.
3. Select a file location for the catalog. The catalog **ExportedKinematics-Catalog.afc** is selected by default.
4. If required, deactivate the check box next to **Replace already existing kinematics in catalog.**.
5. Click on **Finish**. The kinematic systems are exported.
6. If the export was successful, this is indicated by the following message in the **Export kinematics to catalog.** window: **Catalog export successful**. Close the window.

6.24 Editing TOOL and BASE coordinate systems

> ⚠ **WARNING** Changes to the TOOL and BASE coordinate system numbers have no effect on programs that have already been created and positions that have already been taught. Changing the number of a TOOL coordinate system can thus result in unexpected robot motions. If TOOL and BASE coordinate system numbers are changed, it must be noted that all programs containing these TOOL and BASE coordinate systems must also be modified.

6.24.1 Opening tool/base management

Description   The **Tool/base management** editor can be used to create, edit and delete TOOL and BASE coordinate systems. The coordinate systems can also be assigned to a different number by means of Drag&Drop.

Objects can be created, edited and deleted in the **Available tools and bases** area. An object can contain one or more elements. These elements can be assigned to a number in the area of the TOOL and BASE coordinate systems by means of Drag&Drop.
Objects can be exported to a catalog, enabling them to be used in other projects and on other controllers.

Procedure

■ Select the menu sequence **Editors** > **Tool/base management**.
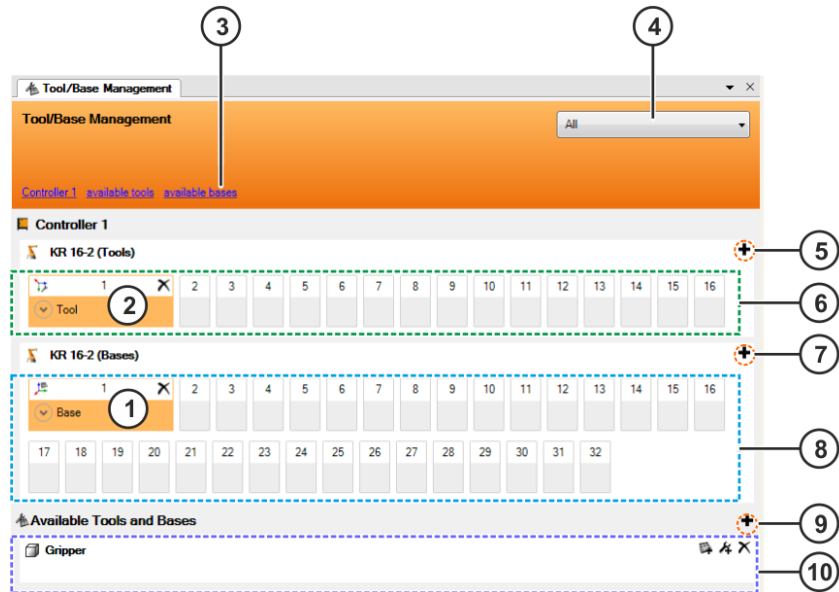
Editor

Fig. 6-8: Tool/base management editor

Item	Description
1	Assigned BASE coordinate system (>>> 6.24.3 "TOOL and BASE coordinate systems" Page 49)
2	Assigned TOOL coordinate system (>>> 6.24.3 "TOOL and BASE coordinate systems" Page 49)
3	The groups of TOOL and BASE coordinate systems that exist in this project are displayed here. Clicking on one group shows this group in the display and hides all other groups.
4	This selection box indicates which groups are currently shown in the display. The selection list contains all the existing groups. The groups can be displayed or hidden using the check boxes.
5	Creates a new TOOL coordinate system on this controller.
6	Number of available TOOL coordinate systems The number can be changed: (>>> 6.16.4 "Changing the number of TOOL and BASE coordinate systems" Page 39)
7	Creates a new BASE coordinate system on this controller.
8	Number of available BASE coordinate systems The number can be changed: (>>> 6.16.4 "Changing the number of TOOL and BASE coordinate systems" Page 39)
9	Creates a new object. An object can contain one or more elements.
10	Objects that have been created or imported in this project The objects can be edited or deleted here. (>>> 6.24.4 "Objects" Page 49)

6.24.2   Configuring tool/base management

Here it is possible to set whether the message displayed when deleting TOOL and BASE coordinate systems should be suppressed.

Procedure

1. Select the menu sequence **Extras** > **Options...**. The **Options** window is opened.
2. On the left of the window, in the **Tool and base management** folder, select the subitem **ToolBaseEditor**.

    The corresponding settings are now displayed on the right of the window.
3. Make the desired setting.
4. Click on **OK** to confirm. The setting is applied.

6.24.3 TOOL and BASE coordinate systems

Description
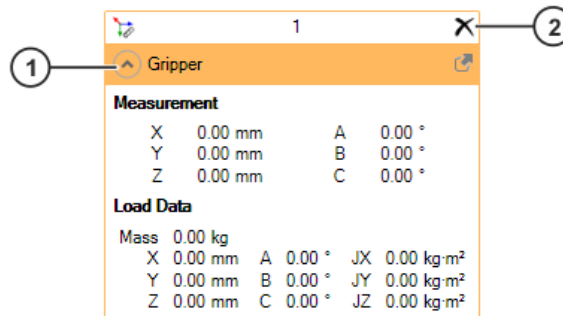
Fig. 6-9: TOOL coordinate system in the Tool/base management editor

1   Displays the coordinate system data
2   Deletes the coordinate system

Procedure

Create TOOL or BASE coordinate system:

1. Click on the ✚ button. A window opens.
2. Enter a name for the coordinate system in the **Name** box.
3. If the data are known: open the **Details** box and enter the measurement data. In the case of TOOL coordinate systems, enter the load data as well.
4. Click on **OK**. The data are saved.

Edit TOOL or BASE coordinate system:

1. Double-click on the coordinate system. A window opens.
2. Modify the name and data as required.
3. Click on **OK**. The data are saved.

Delete TOOL or BASE coordinate system:

- Click on the ✕ button and confirm the message with **Yes**. The coordinate system is deleted.

6.24.4 Objects

Description

Elements of type FRAME (for bases and workpieces on the robot flange) and TCP (for tools on the robot flange and fixed tools) can be created in an object.
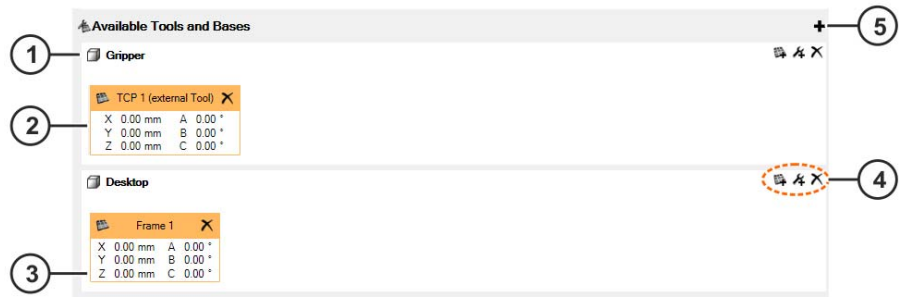
Fig. 6-10: Objects

1	Object
2	Element of type TCP
3	Element of type FRAME
4	Buttons for creating and deleting elements
5	Creates a new object

The type of element as well as its geometric context determine the area to which the element can be assigned.

Tools on the robot flange and fixed tools are assigned to the area of the TOOL coordinate systems. Bases and workpieces on the robot flange are assigned to the area of the BASE coordinate systems.

Geometric context	Element type TCP	Element type FRAME	Area assigned to
FLANGE	Tool on the robot flange	Workpiece on the robot flange	TOOL
WORLD	Fixed tool	Base	BASE
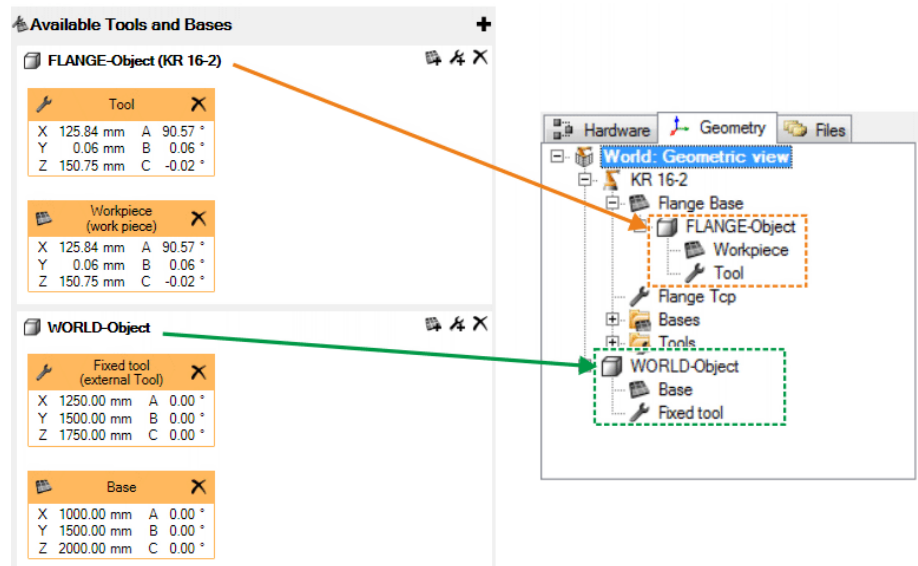
Fig. 6-11: Geometric context
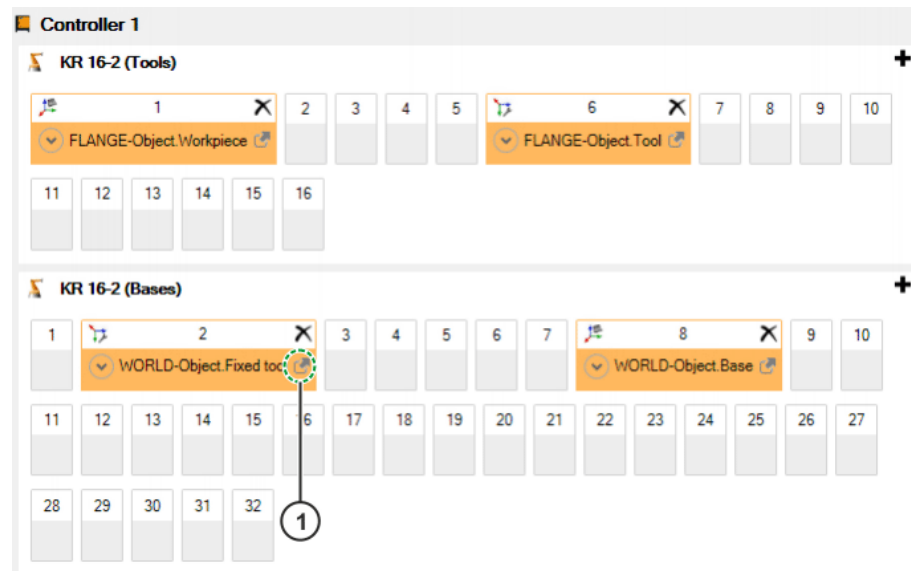
Fig. 6-12: Assigning the elements

Item	Description
1	The icon indicates that the element is from an object. Hovering with the mouse over the icon displays a tool tip with the path to the object.

Procedure

Create object:

1. Click on the ✚ button. A window opens.
2. Enter a name for the object in the **Object** box.
3. Click on **OK**. The object is created.

Add an element to the object:

1. For elements of type TCP: Click on the ⚒ button. A window opens.

   For elements of type FRAME: Click on the ⊞ button. A window opens.
2. Enter a name for the element in the **Name** box.
3. If the data are known: open the **Details** box and enter the measurement data. In the case of elements of type TCP, enter the load data as well.
4. Click on **OK**. The data are saved.

Delete object:

- Click on the ✗ button. The object is deleted.

Edit element:

1. Double-click on the element. A window opens.
2. Modify the name and data as required.
3. Click on **OK**. The data are saved.

Delete element:

- Click on the ✗ button. The element is deleted.

6.24.5 Exporting objects to a catalog

Description

This procedure can be used to export objects to a catalog. In this way, the objects can be used in different projects and on different controllers.

Precondition

- A project is open.

Procedure	1. Right-click on (any) object on the **Geometry** tab in the **Project structure** window and select **Export tool** from the context menu. A window opens.
	2. All cells and objects in the project are displayed in a tree structure. Activate the check boxes next to the objects that are to be exported. Confirm with **Next >**.
	3. Select a file location for the catalog. The catalog **ExportedKinematics-Catalog.afc** is selected by default.
	4. If required, deactivate the check box next to **Replace already existing kinematics in catalog.**.
	5. Click on **Finish**. The objects are exported.
	6. If the export was successful, this is indicated by the following message in the **Export tool to catalog** window: **Catalog export successful**. Close the window.

6.24.6 Importing objects from a catalog

Precondition	■ The catalog with the object is inserted in the project.
Procedure	1. Select the **Geometry** tab in the **Project structure** window.
	2. Depending on the desired geometric context, drag the object either onto the node **World** or onto the **Flange Base** on the **Geometry** tab.
	3. Some of the elements it contains are added directly to the area of the TOOL and BASE coordinate systems. They are, however, not assigned and are thus displayed in red. Assign these elements to the desired numbers by means of Drag&Drop.

6.24.7 Copying and pasting measurement data

Description	The measurement data of a tool or base can be copied and pasted elsewhere. Data can also be copied to the clipboard or pasted from it.

> ℹ The source and target type can be different. For example, the measurement data from a TOOL coordinate system can be inserted into a BASE coordinate system. Only the data also present in the target type are inserted.

The data structure can be created by the user, e.g. on the basis of data from CAD systems. The required layout of the data structure can be seen by copying the data of a coordinate system and pasting it into a text editor.

Example	```
	{X 100.0,Y 0.0,Z 200.0,A 0.0,B 0.0,C 0.0}
	{M 3.0,CM {X 200.0,Y 0.0,Z 150.0,A 0.0,B 0.0,C 0.0},J {X 0.05,Y 0.0325,Z 0.0625}}

Procedure for copying	■ Right-click on the TOOL or BASE coordinate system and select **Copy measurement data**. The data are copied to the clipboard.
Procedure for pasting	■ Right-click on the TOOL or BASE coordinate system and select **Paste**. The data are pasted from the clipboard.

6.25 Defining cells online

6.25.1 Opening the cell definition

Description   Cells can be edited online in the **Cell definition** editor. Cells can be created, edited and deleted. Robot controllers can be assigned to a cell. Furthermore, robot controllers can also be moved to different cells by means of Drag&Drop.

Robot controllers can also be deleted from cells. In this case, the robot controllers are moved back to their original cell. The same applies if a cell containing one or more robot controllers is deleted.

> **i** Deleted cells cannot be restored. If a deleted cell is required again, it must be created again.

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- **Programming and diagnosis** workspace

Procedure
1. Select the desired cells in the **Cell view** window.
2. Select the menu sequence **Editors** > **Cell definition**. The **Cell definition** window opens. The selected cells are displayed and can now be edited.

Editor

Fig. 6-13: Cell definition editor

Item	Description
1	Cell with robot controller
2	Buttons for editing the cell

6.25.2 Configuring a cell definition

Here it is possible to set whether certain messages displayed during the definition of cells should be suppressed.

Procedure
1. Select the menu sequence **Extras** > **Options...**. The **Options** window is opened.
2. On the left of the window, in the **Online workspace** folder, select the subitem **Cell definition editor**.

The corresponding settings are now displayed on the right of the window.

3. Make the desired settings.

4. Click on **OK** to confirm. The settings are saved.

6.25.3   Editing cells

Precondition   ■ The **Cell definition** editor is open.

Procedure   **Create new cell:**

1. Select a robot controller in a cell and drag it into the empty area. A window opens.

2. Enter a name for the cell and confirm it with **OK**. A new cell with this robot controller is created.

Add a robot controller to the cell:

1. Click on the ➕ button in the cell. A window opens.

2. In the list, select the robot controller that is to be added.

3. Click on **OK**. The robot controller is added to the cell.

Rename cell:

1. Click on the 🔳 button in the cell. A window opens.

2. Change the name of the cell and confirm with **OK**. The cell is renamed.

Delete cell:

■ Click on the ✖ button in the cell. The cell is deleted and the robot controllers in this cell receive a new cell of their own. These cells are named by default with the Windows name of the robot controller.

Delete a robot controller in a cell:

1. Position the mouse pointer on the robot controller in the cell. The ✖ button is displayed.

2. Click on the ✖ button. The robot controller is deleted from the cell and receives a new cell of its own. This cells is named by default with the Windows name of the robot controller.

6.26   Option packages

6.26.1   Installing an option package in WorkVisual

Description   Option packages can be installed in WorkVisual. This makes it possible to insert the catalog of the option package in the individual projects, if required. The option package is then available for use in this project.

Advantage: if a project is transferred to a number of robot controllers, the settings relevant to the package only need to be carried out once in WorkVisual and not on each individual robot controller.

On installation, the option package is checked for compatibility and dependencies. If there are conflicts, a message is displayed.

Precondition   ■ The option package is available as a KOP file.

The KOP file is located on the CD of the option package.

■ There is no project open.

Procedure

1. Select the menu sequence **Extras** > **Option package management...**. The **Option package management** window opens.

2. Click on the **Install...** button. The **Select the package to be installed** window opens.

3. Navigate to the path where the option package is located and select it. Click on **Open**.

4. The package is installed. If the KOP file contains device description files, the **Update catalog** window opens and closes during this operation.

   Once the operation has been completed, the package is displayed in the **Installed option packages** area of the **Option package management** window.
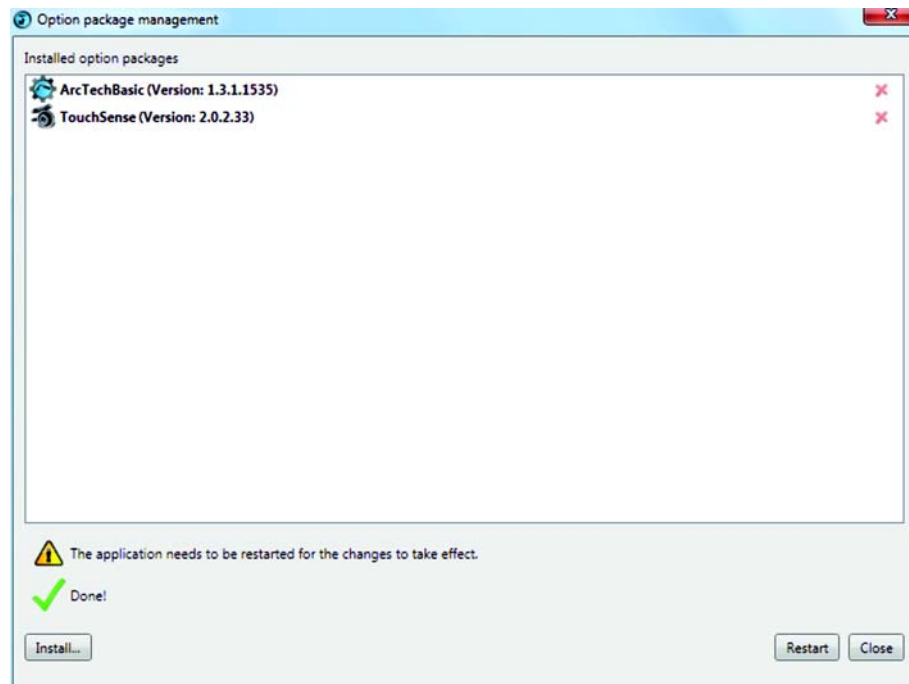
Fig. 6-14: Installed option packages

5. Only if the message *The application must be restarted for the changes to take effect* is displayed:

   - Either: click on the **Restart** button. WorkVisual restarts.
   - Or: close the **Option package management** window and restart WorkVisual later.

6. Only if the message stated in the previous step is NOT displayed: Close the **Option package management** window.

The catalog of the option package is now available under **File** > **Catalog management...**.

If the KOP file contains device description files, these are now available in WorkVisual. It is not necessary to perform a catalog scan.

6.26.2 Updating an option package

Description

Only option packages which do not contain an expansion for WorkVisual (e.g. an additional editor) can be updated. Other option packages must first be uninstalled before the new version can be installed.

For the user, it is not apparent beforehand whether a package is updatable or not. The update process can be started nonetheless. If the previous version has to be uninstalled first, WorkVisual displays a message to this effect.

Precondition

- There is no project open.

Procedure

1. Select the menu sequence **Extras** > **Option package management...**.
   The **Option package management** window opens.

2. Click on the **Install...** button. The **Select the package to be installed** window opens.

3. Navigate to the path where the option package is located and select it. Click on **Open**.

4. One of the following messages is displayed.

   ▪ **Update of {0} (version: {1}) is not possible, as option package contains a WorkVisual add-in.**
     - Uninstall the option package {0} (version: {1})
     - Restart WorkVisual
     - Install the option package {2} (version: {3})

     Confirm the message with **OK**. Do not continue with step 5, but uninstall the package. Then install the new version.

   ▪ **The package is already installed. Would you like to update/downgrade to the package "{0}" with the selected version {1}?**

     Confirm the message with **Yes**.

     The package is installed. If the KOP file contains new device description files, the **Update catalog** window opens and closes during this operation.

5. Close the **Option package management** window.

6.26.3   Uninstalling an option package

Precondition

▪ There is no project open.

Procedure

1. Select the menu sequence **Extras** > **Option package management...**.
   The **Option package management** window opens.

2. In the **Installed option packages** area, click on the red "X" to the right of the name of the package.

3. Only if the option package is now hidden in the **Installed option packages** area: Close the **Option package management** window.

   Uninstallation is now complete. No further steps are necessary.

4. Only if the message *The application must be restarted for the changes to take effect* is displayed:

   ▪ Either: click on the **Restart** button. WorkVisual restarts.

   ▪ Or: close the **Option package management** window and restart WorkVisual later.

If an option package that is used in a project is uninstalled and this project is opened again, the user is prompted to open the catalog for the option package.

If the catalog is not opened, WorkVisual displays the following warning in the message window: **The following option packages of the project are not installed in WorkVisual: {Name}**

6.26.4   Inserting an option package in a project

> **i** This action is not necessary if a device is added to the project from the option package. In this case, the option package is also inserted automatically.

Description

In order to be able to use an option package on the real robot controller, it must be inserted into the project in WorkVisual.

On insertion, the option package is checked for compatibility with the set controller version. If the project already contains option packages, the dependencies in relation to these option packages are also checked. If conflicts are detected during the check, a message is displayed and the insertion is aborted.

> **i** If a project containing an option package is transferred to the robot controller, the procedure differs from the normal deployment procedure. Further information can be found in the section on project deployment.

Precondition
- The catalog of the option package has been added to the project.

Procedure
Project structure
1. Select the **Hardware** tab in the **Project structure** window.
2. Right-click on the **Options** node and select **Add…**.
3. A window opens. Select the catalog **Options**.
4. Select the option package and click on the **Add** button.
5. A window opens. The files being added or modified are indicated in the window.
6. Click on **OK**. The window closes and the project is saved.

The option package is now displayed in the **Options** node.

Procedure Cell configuration
1. Right-click on the robot controller in the **Cell configuration** window and select **Add…**.
2. A window opens. Select the catalog **Options**.
3. Select the option package and click on the **Add** button.
4. A window opens. The files being added or modified are indicated in the window.
5. Click on **OK**. The window closes and the project is saved.

The option package is now displayed in the **Options** node in the **Project structure** window.

6.26.5 Removing an option package from the project

Description
Before an option package is removed, a check is carried out to see if there are dependencies on other option packages. If there are, a message is displayed. Dependent option packages will also be removed automatically.

Precondition
- The robot controller has not been assigned any devices from this option package.

  Any devices assigned must first be removed.

Procedure
1. Select the **Hardware** tab in the **Project structure** window.
2. Expand the **Options** node. All the option packages contained in the project are displayed.
3. Right-click on the package to be removed and select **Delete**.
4. A window opens. The files being removed or modified are indicated in the window.
5. Click on **OK**. The window closes and the project is saved.

The option package is removed from the **Options** node.

6.26.6 Adding a device from an option package to the robot controller

Description
In order to be able to use devices from the option package on the real robot controller, the devices must be inserted into the project in WorkVisual.

A device is a catalog element to which the following configurations can be assigned:

- Configuration of the device
- Bus configuration
- I/O connections
- Long texts

A device is e.g. a weld power source from KUKA.ArcTech.

Precondition
- The catalog of the option package has been added to the project.
- Only if a device with a bus configuration and/or I/O connections is to be inserted:

  No robot controller has been set to active.

Procedure
1. Select the **Hardware** tab in the **Project structure** window.
2. Right-click on the robot controller and select **Add…**.
3. A window opens. Select the catalog of the option package.
4. Select the required device from the list and click on the **Add** button.
5. If there are already configurations assigned to the device, a message is displayed asking if these configurations are to be transferred to the project.

   Select **Yes** or **No** as appropriate.
6. If I/O connections were transferred together with the device, the **Adjust signal connections** window is opened.

   If the signals to which the device is to be connected in accordance with its default settings are already mapped in the current project, this is shown in the **Current conflicts** area.

   (>>> Fig. 6-15)
7. If signals are displayed in the **Current conflicts** area:

   If desired, change the start addresses for the different I/O types until no further conflicts are displayed.
8. Either: Click on **OK**. Any signals remaining in the **Current conflicts** area are now overwritten with the new connections.

   A message is displayed in the message window for each overwritten connection. This makes it easier in the case of later changes.

   Or: Click on **Cancel**. The device is inserted in the **Project structure** window, but no signal connections are adopted.

The device is now displayed underneath the robot controller.

If bus configurations have been accepted together with the device, the device is also displayed under the **Bus structure** node when the robot controller is set back to active.

Adjust signal connections
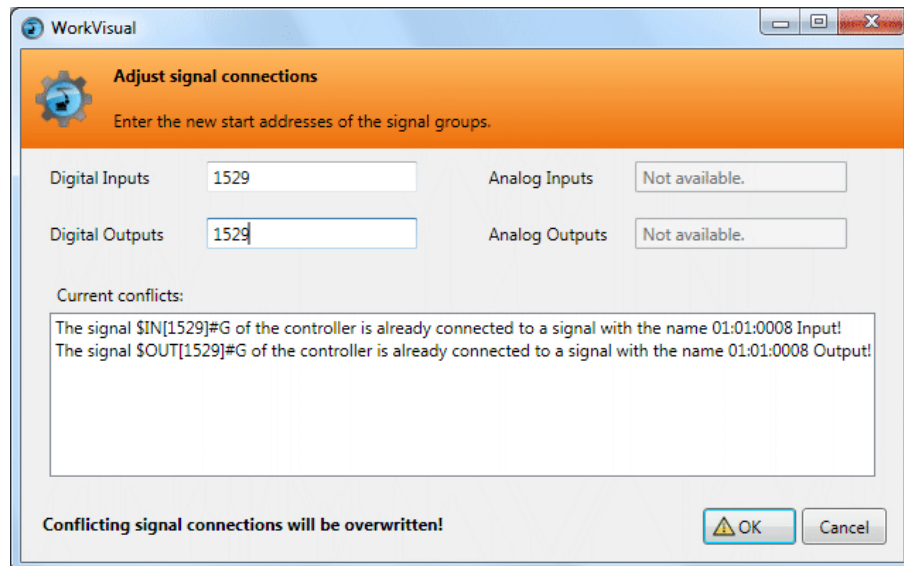
Fig. 6-15: Adjust signal connections – Current conflicts

6.26.7 Exporting a subproject

Description

This procedure enables parts of projects to be exported, e.g. a device and/or I/O connections. Subprojects can be processed further with the KUKA.Option-PackageEditor. The KUKA.OptionPackageEditor is a software product for manufacturers of technology packages and system integrators for the creation of option packages.

Subprojects have the file format WVPS ("WorkVisual Partial Solution").

Precondition

■ A project is open.

■ The robot controller has not been set as the active controller.

Procedure

1. Select the menu sequence **File** > **Import / Export**. A window opens.

2. Select the entry **Export subproject**. The window is now called **Export subproject**.
   Click on **Next**.

3. All the controllers of the project are displayed. Select the controller from which data is to be exported and confirm with **Next**.

4. A tree structure is displayed. In the tree structure, activate the check boxes for the elements that are to be exported. Confirm with **Next**.

5. Select a path for saving the subproject and click on **Finish**. The subproject is exported.

6. If the export was successful, this is indicated by the following message in the **Export subproject** window: **Partial export successful**.
   Close the window.

6.27 Changing predefined properties of WorkVisual

6.27.1 Configuring booting and saving characteristics

Procedure

1. Select the menu sequence **Extras** > **Options...**. The **Options** window is opened.

2. On the left of the window, in the **Project handling** folder, select the subitem **Project handling**.
   The corresponding settings are now displayed on the right of the window.

3. Make the desired settings.

4. Confirm the changes made with **OK**.

Description   Subitem **Project handling**:

Parameter	Description
Project folder directory:	A directory can be specified here in which the projects are to be saved by default.
Catalog directory:	This specifies the directory in which the catalogs are saved. If the catalogs are moved to a different directory, the directory given here must be changed.
Start application	Here it is possible to specify whether a new project, the most recently opened project or no project is offered for opening when WorkVisual is started.

6.27.2   Configuring keyboard shortcuts

Procedure

1. Select the menu sequence **Extras** > **Options**. The **Options** window is opened.

2. On the left of the window, in the **Environment** folder, select the subitem **Keyboard**.

    The corresponding settings are now displayed on the right of the window.

3. In the **Commands** box, select the command for which a keyboard shortcut is to be defined or changed.

    The contents of the **Command** box can be filtered:

    Enter a term in the box **List only commands with the following content**. Only commands whose names contain this term are now displayed in the **Command** box.

4. Place the cursor in the **New shortcut** box and confirm the desired key combination (or individual key) on the keyboard. Examples: **F8** or **Ctrl+W**

    The keyboard shortcut is displayed in the **New shortcut** box.

5. Click on **Assign**.

6. Confirm the change with **OK**.

    Or: If the keyboard shortcuts are already assigned, a request for confirmation is displayed:

    ■ If the keyboard shortcut is to be assigned to the new command, select **Yes** and confirm the changes with **OK**.

    ■ If the keyboard shortcut is to remain assigned to the old command, select **No**.

        Close the window by selecting **Cancel**. Or delete the keyboard shortcut from the **New shortcut** box using the **Esc** key, and enter a different one.

6.27.3   Changing the user interface language

Description   Which languages are available here depends on which languages were installed with WorkVisual.

Procedure

1. Select the menu sequence **Extras** > **Options**. The **Options** window is opened.

2. On the left of the window, select the **Localization** folder.

    The corresponding settings are now displayed on the right of the window.

3. Select the desired language from the **Language** box. Confirm with **OK**.

4. Close and restart the application.

6.27.4 Configuring additional editors

Description

Additional editors are configured using this procedure. These editors can be used to open, compare and merge files. The first editor in the list is the default editor.

> Various editors are already preconfigured by default. These are not included in the scope of supply of WorkVisual however. They can therefore only be used if they have been installed beforehand.

Procedure

1. Select the menu sequence **Extras** > **Options...**. The **Options** window is opened.

2. On the left of the window, in the **Additional editors** folder, select the desired subitem.

   The corresponding settings are now displayed on the right of the window.

3. Click on the **+** button at the top right. A new program is added.

4. Enter the name of the program in the **Name** box.

5. In the **File extension** box, enter the file formats that are to be opened with the program.

6. Click on the **…** button next to the **Program** box and select the EXE file of the program.

7. Optionally: Enter one or more parameters in the **Parameter** box. The **Right arrow** button displays an explanation of the parameters.

> Further information about calling an external editor using the command line can be found in the vendor's documentation for the external editor.

8. Confirm the changes made with **OK**.

9. If the editor is to be used as the default editor: Use the **Up arrow** button to move the editor to the top of the list.

6.28 Print functions

The procedure described here can be used to print the following:

- I/O connections
- Long texts
- Safety configuration

Procedure

1. Select the menu sequence **File** > **Print**. The **Print** window is opened.

2. Select the desired printer in the **Printer** area. If required, change the printer settings.

3. In the **Document** area, select the relevant checkboxes to indicate what should be printed.

4. If required, first display the print preview: To do this, click on the **Preview** button.

   Close the print preview again.

5. Click on the **Print** button to start printing.

It is also possible to print directly from the print preview, via the button with the printer symbol. In this case, the standard printer is used. Changes to the printer settings are not possible.

Description   **Print** window, **Documents** area:

Check box	Description
Global	This function is not yet assigned.
Cell	If this checkbox is activated, all the robot controllers belonging to this cell are automatically activated. Individual controllers can be deactivated again manually.
Controller […]	If this checkbox is activated, all the documents belonging to this robot controller are automatically selected for printing. Individual documents can be deactivated again manually.
Documents:	
Connection list	Prints the I/O connections defined in the **I/O Mappings** window.
Long texts	If long texts have been defined in various languages, it is also possible to select which languages should be printed.
Safety configuration	The printout contains a date and signature box and can be used as a sign-off sheet for the safety acceptance procedure.

7 Safety configuration

7.1 Safety configuration in WorkVisual

The safety configuration in WorkVisual comprises the following areas:

Area	Description
Local safety configuration	The local safety configuration comprises the parameters in the **Local safety configuration** window. The parameters can be edited.
Safety-relevant communication parameters	These include the parameters relevant to safe communication within a robot network.
	The safety-relevant communication parameters cannot be displayed or edited directly. However, various actions in WorkVisual have an effect on the safety-relevant communication parameters, e.g. if a RoboTeam is configured.

When a project is transferred to the real robot controller, the entire safety configuration is always transferred at the same time.

7.2 Editing the local safety configuration

Description
A newly added robot controller is without a local safety configuration in WorkVisual. A robot controller without a local safety configuration can be recognized by the fact that the text of the **Safety controller** node on the **Hardware** tab in the **Project structure** window is in italics: *Safety controller*

The robot controller is automatically assigned a local safety configuration in WorkVisual when the **Local safety configuration** window is opened. The robot controller is assigned a local safety configuration during code generation, if none has been assigned already.

The local safety configuration can be edited in WorkVisual. The changes always apply to the robot controller which is currently set as active.

Precondition
- The robot controller has been set as the active controller.
- A robot has been assigned to the robot controller.

Procedure
1. Double-click on the node **Safety controller** on the **Hardware** tab in the **Project structure** window. The **Local safety configuration** window is opened.
2. If a safety option, e.g. SafeOperation, is used:
   a. On the **General** tab, select the **Global parameters** area.
   b. Here, activate the **Safe monitoring** check box. Only then can the monitoring functions be edited.
3. Edit the parameters of the safety configuration as required.
4. Close the **Local safety configuration** window.

7.3 Parameters of the local safety configuration

> **i** The standard parameters are described here. Information about parameters which refer to a particular safety option can be found in the documentation for the specific safety option.

7.3.1   "General" tab (8.2)

Hardware options

Parameter	Description
Customer interface	Select here which interface is used: ■ **ProfiSafe** ■ **SIB** ■ **SIB, Extended SIB** ■ **SIB with operating mode output** ■ **SIB with operating mode output, Extended SIB** This option is available with System Software version 8.2.4 or higher. The following interfaces are available with the controller variant "KR C4 compact": ■ **ProfiSafe** ■ **X11**
Input signal for peripheral contactor (US2)	■ **Deactivated**: The peripheral contactor is not used. (default) ■ **By external PLC**: The peripheral contactor is switched by an external PLC via input US2. ■ **By KRC**: The peripheral contactor is switched in accordance with the motion enable. If motion enable is present, the contactor is energized. **Notes:** ■ For robot controllers with peripheral contactors and the "UL" option, this parameter must be set to **By KRC**. ■ For robot controllers with no peripheral contactors, this parameter has no effect.
Operator safety acknowledgement	If the "Operator Safety" signal is lost and set again in Automatic mode, it must be acknowledged before operation can be continued. ■ **By acknowledgement button**: Acknowledgement is given e.g. by an acknowledgement button (situated outside the cell). Acknowledgement is communicated to the safety controller. The safety controller re-enables automatic operation only after acknowledgement. ■ **External unit**: Acknowledgement is given by the system PLC.

Change log   Every modification to the local safety configuration and every saving operation are automatically logged. The log is displayed here.

Machine data   The machine data of the safety controller are displayed here.

> ℹ️ It is not necessary to press the **Import machine data** button. There are currently no applications in which this is necessary.

Communication parameters
The Profinet Safety ID is displayed here. This is required if the robot controller is used as a PROFINET device. The ID can be changed when **ProfiSafe** is selected as the customer interface.

7.3.2 "General" tab (8.3)

Hardware options

Parameter	Description
Customer interface	Select here which interface is used: ■ **Automatic** ■ **SIB with operating mode output**
Input signal for peripheral contactor (US2)	■ **Deactivated**: The peripheral contactor is not used. (Default) ■ **By external PLC**: The peripheral contactor is switched by an external PLC via input US2. ■ **By KRC**: The peripheral contactor is switched in accordance with the motion enable. If motion enable is present, the contactor is energized. **Notes:** ■ For robot controllers with peripheral contactors and the "UL" option, this parameter must be set to **By KRC**. ■ For robot controllers with no peripheral contactors, this parameter has no effect. The system variable $US2_VOLTAGE_ON indicates the status of the peripheral voltage US2: ■ TRUE: Voltage is switched on. ■ FALSE: Voltage is switched off.
Operator safety acknowledgement	If the "Operator Safety" signal is lost and set again in Automatic mode, it must be acknowledged before operation can be continued. ■ **By acknowledgement button**: Acknowledgement is given e.g. by an acknowledgement button (situated outside the cell). Acknowledgement is communicated to the safety controller. The safety controller re-enables automatic operation only after acknowledgement. ■ **External unit**: Acknowledgement is given by the system PLC.

Change log
Every modification to the local safety configuration and every saving operation are automatically logged. The log is displayed here.

Machine data
The machine data of the safety controller are displayed here.

> ℹ️ It is not necessary to press the **Import machine data** button. There are currently no applications in which this is necessary.

Communication parameters
The Profinet Safety ID is displayed here. This is required if the robot controller is used as a PROFINET device. The ID can be changed when the PROFINET Device Stack is activated.

7.3.3 "Axis monitoring" tab (8.3)

Editable parameters
The following parameters can be set for each axis. It is not generally necessary to change the default values, however.

Parameter	Description
Braking time	Duration of the monitored axis-specific braking ramp for safety stop 1 and safety stop 2 Default: 1,500 ms (>>> 7.3.3.1 "Parameter: Braking time" Page 66)
Maximum velocity T1	Maximum velocity in T1 ■ Rotational axes: **1.00 … 100.00 °/s** Default: 30 °/s ■ Linear axes: **1.00 … 1,500.00 mm/s** Default: 250 mm/s This parameter enables a servo gun, for example, to be calibrated in T1 with a higher velocity than 250 mm/s. **Note:** The Cartesian velocities at the flange and at the TCP are monitored independently of this parameter and cannot exceed 250 mm/s.
Position tolerance	Tolerance for standstill monitoring in the case of safe operational stop. The axis may still move within this tolerance when a safe operational stop is active. ■ Rotational axes: **0.001 … 1 °** Default: 0.01 ° ■ Linear axes: **0.003 … 3 mm** Default: 0.1 mm

7.3.3.1 Parameter: Braking time

Description

If a safety stop 1 or 2 occurs, the safety controller monitors the braking process. Among other things, it monitors whether the axis-specific velocity remains below the braking ramp. If the velocity is too high, i.e. if the braking ramp is violated, then the safety controller triggers a safety stop 0.

The braking ramp can be specified using the parameter **Braking time**.

> The parameter **Braking time** modifies the braking ramp. It does not modify the actual time required by the kinematic system for braking.

The parameter **Braking time** has no effect in T1, since it refers to the axis-specific monitoring. In T1, however, there is another (non-configurable) monitoring function for the Cartesian velocity on the flange. This is stricter, which is why the axis-specific monitoring has no effect.

> ⚠ **WARNING** Only alter the default time if it is necessary to do so. This might be required, for example, in the case of very heavy machines and/or very heavy loads as these cannot stop within the default time.
> The safety recovery technician must check whether and to what extent the **Braking time** value needs to be modified in each specific application. He must also check whether the modification makes additional safety measures necessary, e.g. installation of a gate lock.

> ℹ️ **Braking time** can be entered separately for each axis; at the moment of braking, however, the value used for all axes is always the highest value entered.
>
> Recommendation: for greater transparency, enter the same value for all axes.

Value increased

If the value Braking time is increased, this has the following consequences:

The braking ramp becomes longer and flatter, i.e. monitoring is now less strict. The same braking process is now less likely to violate the braking ramp than before.
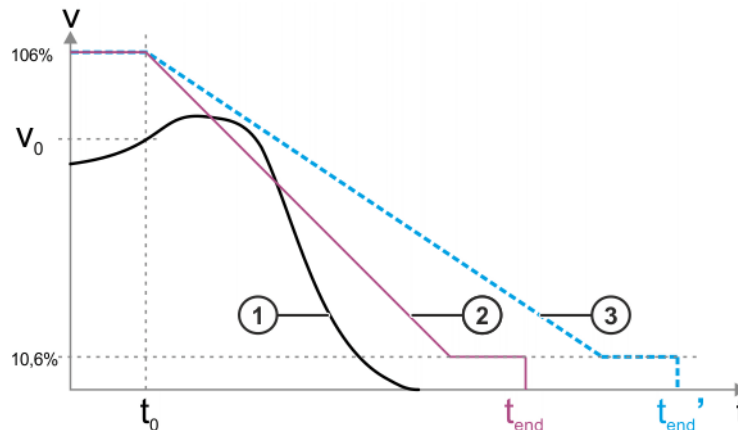
Fig. 7-1: Example: value is increased

1	Velocity profile during braking (example)
2	Braking ramp (original value **Braking time**)
3	Braking ramp (higher value of **Braking time**)
v_0	Velocity of the kinematic system at the moment when braking begins
t_0	Start time of the braking ramp
t_{end}	End of the braking ramp
t_{end}'	End of the braking ramp with a higher value for **Braking time**

The start velocity of the axis-specific braking ramp is always 106% of the rated speed of the axis. The ramp falls to 10.6%. The velocity then remains constant for 300 ms, before falling to 0%.

Value reduced

If the value "Braking time" is reduced, this has the following consequences:

The braking ramp becomes shorter and steeper, i.e. monitoring is now stricter. The same braking process is now more likely to violate the braking ramp than before.

7.4 Importing the safety configuration (SCG import)

> ⚠ **WARNING** After importing a safety configuration or parts thereof, the safety configuration must be checked! If this is not done, this can lead to the possibility of the robot being operated with incorrect data when the project is transferred to the real robot controller. Death to persons, severe injuries or considerable damage to property may result.
> To avoid this, the current safety configuration on the robot controller can be compared with the safety configuration in the SCG file.
> (>>> 7.8 "Comparing the safety configuration" Page 71)

Precondition
- The robot controller has been set as the active controller.

Procedure
1. Select the menu sequence **File** > **Import / Export**. A window opens.
2. Select **Import local safety configuration** and click on **Next**.
3. Navigate to the path where the SCG file is located and select it. Click on **Open**.
4. Click on **Finish**.
5. If the configuration was imported successfully, this is indicated by a message. Close the window.

7.5 Exporting the safety configuration (SCG export)

Precondition
- The robot controller has been set as the active controller.

Procedure
1. Select the menu sequence **File** > **Import / Export**. A window opens.
2. Select **Export local safety configuration** and click on **Next**.
3. Click on **Browse…** and specify a directory.
4. Specify a file name, select the file type SCG and click on **Save**.
5. Click on **Finish**.
6. If the configuration was exported successfully, this is indicated by a message. Close the window.

7.6 Importing the safety configuration (XML import)

> ⚠ **WARNING** After importing a safety configuration or parts thereof, the safety configuration must be checked! If this is not done, this can lead to the possibility of the robot being operated with incorrect data when the project is transferred to the real robot controller. Death to persons, severe injuries or considerable damage to property may result.

Description
Certain parts of the local safety configuration can be imported as an XML file. These are:
- Cell configuration
- Monitoring spaces (Cartesian spaces and/or axis spaces)
- Properties of the tools
- Global parameters

In order to generate an XML file for importing, the user has the following options:
- Export the current safety configuration of the robot controller to an XML file and edit it. In this way it is possible to ensure that the format of the XML file is correct for a subsequent import (>>> 7.7 "Exporting the safety configuration (XML export)" Page 70).

■ Generate the XML file on the basis of the XML schema C:\Program Files (x86)\KUKA\WorkVisual [*Version number*]\Schemes\SafetyConfigImport.xsd, e.g. using a script programmed by the user.

The following points must be observed when editing the XML files:

■ The XML schema defines the structure of the XML file for the import. For individual parameters, the XML schema allows higher values than the installed version of the safety option.

■ Parameters and values that are not supported by the current safety option are not imported. During the import, WorkVisual generates a message to this effect.

> **i** It is also possible to import safety configurations in the system software. Information about this can be found in the documentation of the safety options (e.g. SafeOperation).

Precondition
■ The robot controller has been set as the active controller.
■ The safety option SafeOperation or SafeRangeMonitoring is used.

Procedure
1. Save the project. (Do not close.)
2. Select the menu sequence **File** > **Import / Export**. A window opens.
3. Select **Import local safety configuration** and click on **Next >**.
4. Click on **Search…**. Navigate to the path where the XML file is located and select it. Click on **Open**.
5. Click on **Next >**. In the background, the window **Local safety configuration** is opened, if not already open.
6. If errors are present: error messages are displayed in the Import window. The import cannot be carried out until these errors have been eliminated. Rectify the errors in the XML file, repeat the XML import and save the safety configuration.
7. If no errors are present: the differences between the existing values and those to be imported are displayed in the Import window before the import is carried out. (>>> Fig. 7-2).
8. Check all values.
   If not all of the required safety functions are configured correctly, or if the wrong XML file was selected, cancel the XML import.
   ■ Rectify the error in the XML file and repeat the XML import.
   ■ OR: Select the correct XML file and repeat the XML import.
9. Click on **Import**. The data are now imported.
10. When the import is finished, this is indicated by the following message: **The local safety configuration was imported successfully.**
    Close the window.
11. Check the safety configuration. The modified values are displayed in blue in the **Local safety configuration** window.
12. Save the project to accept the imported data.

> **i** The imported data are only accepted when the project is saved. This also means that imported data can be discarded by closing the project without saving.

Parameter comparison view
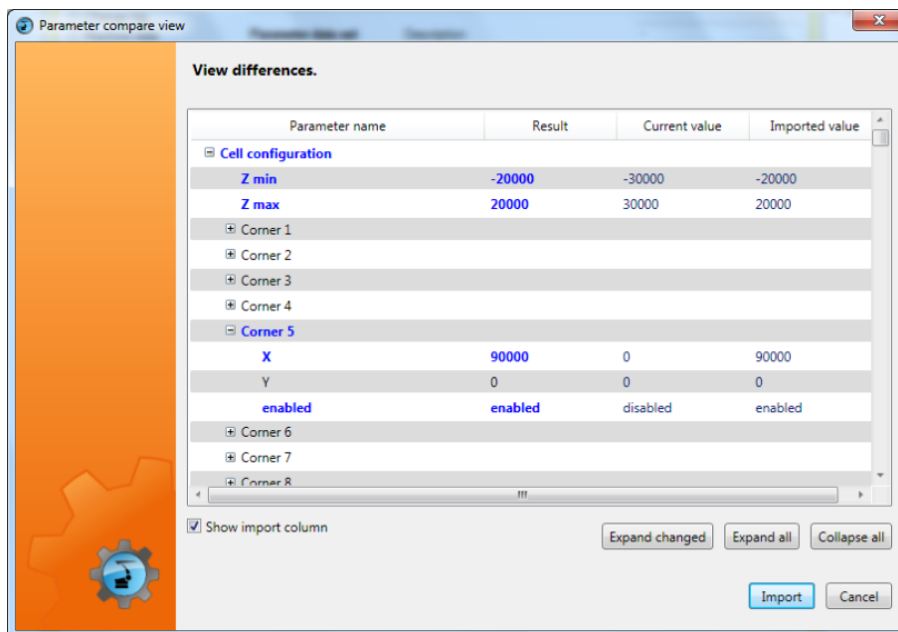
Fig. 7-2: Example: Displaying the differences

Color	Meaning
Blue	With this element (or its child elements), the existing value differs from the value to be imported.
Black	With this element (including all its child elements), the existing value is identical to the value to be imported.

If the **Show import column** check box is active, the **Import value** column is displayed. The values contained in the XML file are displayed in this column.

7.7   Exporting the safety configuration (XML export)

Description   Certain parts of the local safety configuration can be exported as an XML file. These are:

- Cell configuration
- Monitoring spaces (Cartesian spaces and/or axis spaces)
- Properties of the tools
- Global parameters

The XML file always contains all the parameters which are contained in the exported parts of the safety configuration.

Exporting is always possible, irrespective of whether a safety option is installed or not. However, an export only makes sense if a safety option is installed.

The current safety configuration of the robot controller is exported. If the safety configuration contains unsaved changes, these are also exported.

If invalid values are entered in the safety configuration, the export is aborted with an error message (plausibility error).

> **i** It is also possible to export safety configurations in the system software. Information about this can be found in the documentation of the safety options (e.g. SafeOperation).

Precondition   ■   The robot controller has been set as the active controller.

Procedure
1. Select the menu sequence **File** > **Import / Export**. A window opens.
2. Select **Export local safety configuration** and click on **Next**.
3. Click on **Browse…** and specify a directory.
4. Specify a file name, select the file type XML and click on **Save**.
5. Click on **Finish**.
6. If the configuration was exported successfully, this is indicated by a message. Close the window.

7.8 Comparing the safety configuration

Description
To avoid importing an undesired safety configuration, the current safety configuration on the robot controller can be compared with the safety configuration in the SCG file.

Procedure
1. Load the active project from the robot controller in WorkVisual.
   (>>> 13.7 "Loading the project from the robot controller" Page 144)
2. Import the safety configuration (SCG file).
   (>>> 7.4 "Importing the safety configuration (SCG import)" Page 68)
3. Export parts of the safety configuration (XML file) out of the project.
   (>>> 7.7 "Exporting the safety configuration (XML export)" Page 70)
4. Close the project. (Do not transfer to the robot controller.)
5. Reload the active project from the robot controller.
6. Import the parts of the safety configuration from step 3 into the project.
   (>>> 7.6 "Importing the safety configuration (XML import)" Page 68)
7. Compare the parts of the safety configuration with one another.
8. If the parts of the safety configuration are as required, import the safety configuration (SCG file) and transfer it to the robot controller.

7.9 Resetting the safety configuration

Description
This procedure can be used to reset the safety configuration. This may be necessary, for example, if the system software is to be changed to a version whose safety configuration is not compatible with the current one. In this way, only the safety configuration is recreated – all other settings in the project are retained.

When the safety configuration is reset, all settings are reset to the default settings and the project is automatically saved. This process cannot be undone.

Precondition
- The robot controller has not been set as the active controller.

Procedure
1. Right-click on the node **Safety controller** on the **Hardware** tab in the **Project structure** window.
2. Select **Reset** in the context menu.
3. Answer the request for confirmation with **Yes**. The safety configuration is reset.

8 Field bus configuration

8.1 Overview of field buses

The following field buses can be configured with WorkVisual:

Field bus	Description
PROFINET	An Ethernet-based field bus. Data exchange is carried out on a client-server basis. PROFINET is installed on the robot controller.
PROFIBUS	Universal field bus which enables communication between devices from different manufacturers without special interface adaptations. Data exchange is carried out on a master-slave basis.
DeviceNet	CAN-based field bus used primarily in automation technology. Data exchange is carried out on a master-slave basis.
EtherNet/IP	An Ethernet-based field bus. Data exchange is carried out on a client-server basis. EtherNet/IP is installed on the robot controller.
EtherCAT	An Ethernet-based field bus suitable for real-time requirements.
VARAN slave	Field bus that can be used to establish communication between a VARAN controller and a KR C4 controller.

For configuration of a field bus, the documentation of the field bus is required.

8.2 Field bus set-up

8.2.1 Overview: field bus set-up

Step	Description
1	Install the device description files on the PC. (>>> 6.8 "Importing device description files" Page 31)
2	Insert the DTM Catalog in the **Catalogs** window. (>>> 6.9.2 "Inserting a catalog in a project" Page 33)
3	Insert the field bus master in the project. (>>> 8.2.2 "Inserting a field bus master in a project" Page 74)
4	Configure the field bus master. (>>> 8.2.3 "Configuring the field bus master" Page 74)
5	Add the devices to the bus, i.e. insert them under the field bus master. (>>> 8.2.4 "Inserting devices manually into the bus" Page 74) Or: (>>> 8.2.6 "Automatically inserting devices into the bus (Bus Scan)" Page 75)
6	Configure the devices. (>>> 8.2.5 "Configuring devices" Page 75)

Step	Description
7	Edit the field bus signals
	(>>> 8.3 "Editing field bus signals" Page 76)
8	The bus I/Os can now be mapped.
	(>>> 8.4 "Mapping the bus I/Os" Page 81)

8.2.2 Inserting a field bus master in a project

Precondition
- The device description files are inserted in the DTM Catalog (Catalog Scan).
- The robot controller has been added and set as active.

Procedure
1. Expand the tree structure on the **Hardware** tab in the **Project structure** window until the **Bus structure** node is visible.
2. Select the required field bus master in the **DTM Catalog** window and drag it onto the **Bus structure** node.

8.2.3 Configuring the field bus master

Precondition
- The field bus master is inserted in the project.
- The robot controller has been set as the active controller.

Procedure
1. Right-click on the field bus master on the **Hardware** tab in the **Project structure** window.
2. Select **Settings…** from the context menu. A window opens with device data.
3. Set the data as required and save with **OK**.

> **NOTICE** The following address ranges are used by default by the robot controller for internal purposes. IP addresses from this range must not therefore be assigned by the user.
> - 192.168.0.0 … 192.168.0.255
> - 172.16.0.0 … 172.16.255.255
> - 172.17.0.0 … 172.17.255.255

> **i** Information about settings for particular bus systems can be found in the documentation for these bus systems.

8.2.4 Inserting devices manually into the bus

> **NOTICE** The inserted device must correspond to the actual device used in reality. Substantial damage to property may otherwise result.

Precondition
- The devices are inserted in the DTM Catalog.
- The field bus master is inserted in the bus structure.
- The robot controller has been set as the active controller.

Procedure
1. Expand the tree structure on the **Hardware** tab in the **Project structure** window until the field bus master is visible.
2. Select the required device in the DTM Catalog and drag it onto the field bus master.

3. If necessary, repeat step 2 for further devices.

8.2.5 Configuring devices

Precondition
- The device has been added to the bus.
- The robot controller has been set as the active controller.

Procedure
1. Right-click on the device on the **Hardware** tab in the **Project structure** window.
2. Select **Settings…** from the context menu. A window with the device data is opened.
3. Set the data as required and save with **OK**.

> **NOTICE** The following address ranges are used by default by the robot controller for internal purposes. IP addresses from this range must not therefore be assigned by the user.
> - 192.168.0.0 … 192.168.0.255
> - 172.16.0.0 … 172.16.255.255
> - 172.17.0.0 … 172.17.255.255

> **i** Information about settings for particular bus systems can be found in the documentation for these bus systems.

8.2.6 Automatically inserting devices into the bus (Bus Scan)

Description
The bus scan is available for Interbus and EtherCAT.

The bus devices can be inserted automatically. For this, the user can start a scan in WorkVisual to determine which devices are connected to the real bus. The corresponding devices are then inserted automatically into the bus structure in WorkVisual.

In contrast to manual insertion, this procedure is faster and less susceptible to error.

Precondition
- The devices are inserted in the DTM Catalog.
- The field bus master is inserted in the bus structure.
- The robot controller has been set as the active controller.
- Network connection to the real robot controller
- The devices are connected to the real robot controller.

> **i** There are further preconditions depending on the bus system used. Information about this is contained in the documentation for the different bus systems.

Procedure
1. Expand the tree structure of the robot controller on the **Hardware** tab in the **Project structure** window.
2. Right-click on the field bus master. Select the option **Scan Topology…** and then select a channel. The **Topology Scan Wizard** window is opened.
3. Click on **Continue** to start the scan. When the scan is completed, WorkVisual displays all the devices found in the window on the left. Each device is represented by a number (= product code).
4. Select a device. In the window on the right, WorkVisual displays a list of the device description files which have the same product code.

5. If the list contains a number of device description files, scroll down the list and check if the file of the device that is actually used is selected. If a different file is marked, select the option **Manual selection** and select the correct file.

6. Repeat steps 4 and 5 for all devices shown.

7. Click on **Continue** to confirm the assignment.

8. Click on **Finish** to assign the devices to the field bus master.

8.3 Editing field bus signals

Description Field bus signals can be edited in WorkVisual. For example, the signal width can be changed or the byte order can be swapped.

Precondition ■ The field bus devices are configured.

Procedure
1. Select the device on the **Field buses** tab in the **I/O Mapping** window.

2. In the **I/O Mapping** window, click on the **Edit signals at the provider** button. The **Signal Editor** window is opened. All inputs and outputs of the device are displayed.

   (>>> 8.3.1 "Signal Editor" Page 76)

3. Edit the signals as required.

4. Click on **OK** to save the changes and close the **Signal Editor** window.

8.3.1 Signal Editor

In the Signal Editor, the inputs of the selected device are displayed on the left and the outputs on the right.
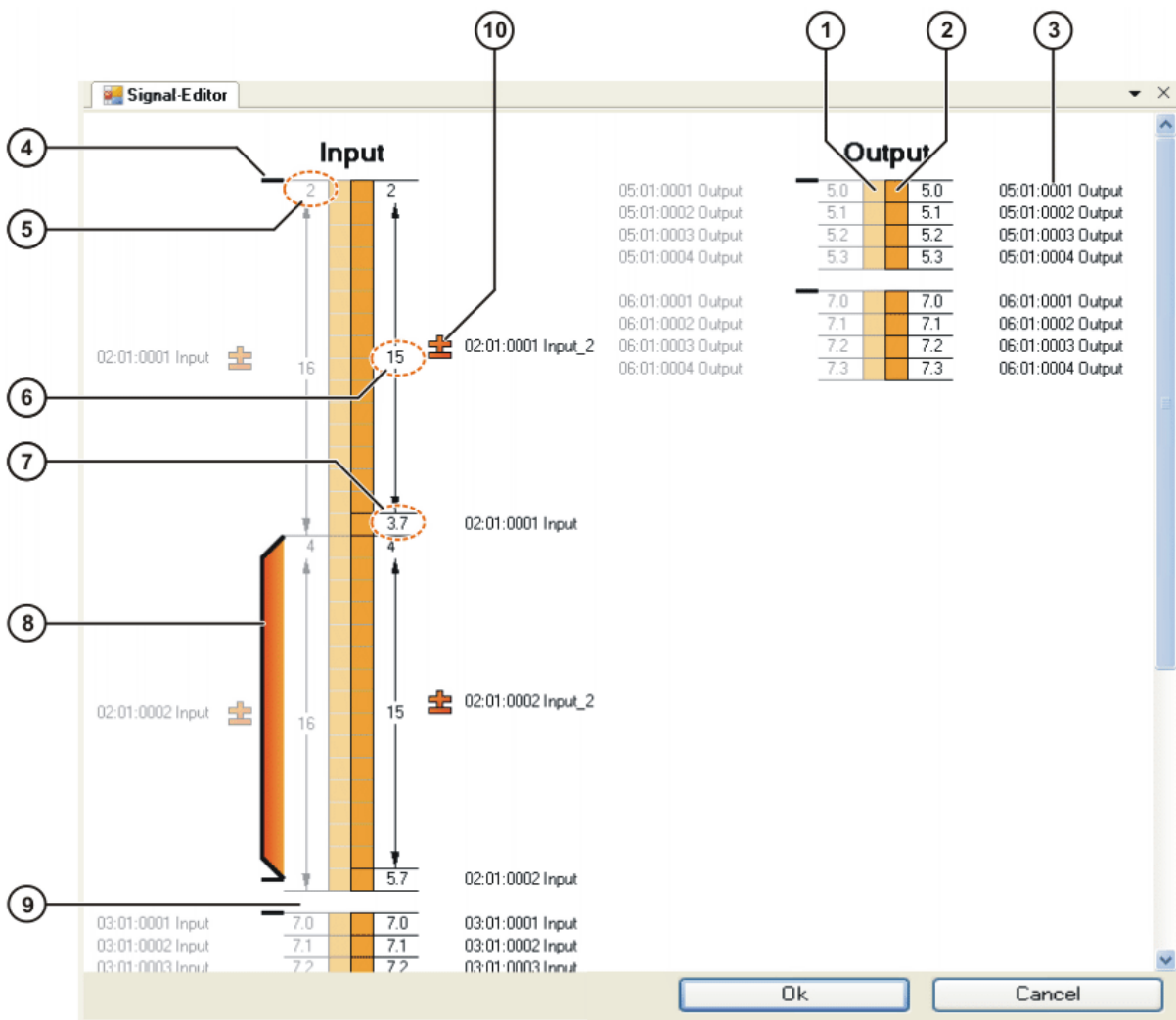
Fig. 8-1: Signal Editor

Item	Description
1	The left-hand column displays the original configuration of the inputs or outputs. Each box represents 1 bit.
2	The right-hand column can be edited and always displays the current configuration of the inputs or outputs. Each box represents 1 bit.
3	Signal name (>>> 8.3.5 "Changing signal names" Page 80)
4	Start mark for swapping (>>> 8.3.3 "Swapping signals (reversing the byte order)" Page 78)
5	Address at which this signal starts
6	Signal width (>>> 8.3.2 "Changing the bit width of signals" Page 78)
7	Address to which this bit belongs, and number of the bit
8	The bar indicates that the byte order has been swapped.
9	Boundary between memory segments
10	Data type of this signal (>>> 8.3.4 "Changing the data type" Page 80)

8.3.2   Changing the bit width of signals

Description

This procedure can be used to change the width of signals. Signals can be split or grouped. It is also possible to split signals more than once.

The signal boundaries can be moved only as far as the boundaries of the memory segments. Signal boundaries cannot be moved beyond the boundaries of swapped ranges.

Bits that are being edited are indicated in red.

Precondition

- The Signal Editor is open.
- The signals that are to be edited are not mapped.

Procedure

Moving a signal boundary:

1. In the right-hand column, position the mouse pointer over the boundary line between 2 signals. The mouse pointer changes into a vertical double arrow.
2. Click and hold down the mouse button and move the mouse pointer up or down. The boundary line is moved.
3. Drag the boundary line to the desired position and let go.

This procedure can be used to reduce a signal down to the size of 1 bit.

Splitting a signal:

1. In the right-hand column, position the mouse pointer over a bit.
2. Click and hold down the mouse button and move the mouse pointer up or down. A line is displayed over the output bit.
3. Drag the mouse pointer to another bit and let go. A line is also displayed over this bit. The two lines are the boundaries of the new signal.

Grouping signals:

1. In the right-hand column, position the mouse pointer over the first (or last) bit of a signal.
2. Click and hold down the mouse button and move the mouse pointer down (or up).
3. Drag the mouse pointer across a signal boundary to another signal boundary and let go. The intervening signal boundary disappears. The signals have been grouped.

8.3.3   Swapping signals (reversing the byte order)

Description

The byte order of signals can be swapped. It is possible to swap 2, 4 or 8 bytes at once. Sub-ranges of signals cannot be swapped. Furthermore, it is not possible to swap across the boundaries of memory segments.

The bits within a byte always remain unchanged.

> **i** The (V)KR C4 robot controller uses the Intel data format. Field bus signals in Motorola format must be converted to Intel. This is done by means of swapping.

> It is not always clear from the manufacturer's data sheets whether a signal needs to be swapped or not. The signals of Siemens devices generally need to be swapped. The following procedure can provide information about whether it is necessary to swap an input:
>
> 1. Change the voltage on the input slowly and evenly.
> 2. Observe the values for this signal in the **Analog I/O** window on the KUKA.smartHMI.
>
> If the values change abruptly and unevenly or in different directions, this is an indication that swapping is necessary.

It makes a difference whether a range is swapped as a whole or in parts:

Fig. 8-2: Example 1: Swapping the byte order

Item	Description
1	Original order
2	Result of swapping the bytes in two groups (i.e. the first two bytes are swapped and the second two bytes are swapped separately.)

Fig. 8-3: Example 2: Swapping the byte order

Item	Description
1	Original order
2	Result of swapping all the bytes at once

Precondition

- The Signal Editor is open.

Procedure

1. Position the mouse pointer over a start mark for swapping. The mouse pointer changes into a vertical double arrow.
2. Click and hold down the mouse button. Move the mouse pointer downwards to the signal boundary.
3. A bar is displayed.
   - Either: Release the mouse button. The byte order has now been swapped.
   - Or: If a larger range is to be swapped, keep moving the mouse pointer without letting go. A longer bar is displayed. Release the mouse button. The byte order has now been swapped.

   An end mark for swapping is displayed.

To undo the swapping:

1. Position the mouse pointer over an end mark for swapping. The mouse pointer changes into a vertical double arrow.
2. Click and hold down the mouse button. Move the mouse pointer up towards the start mark.
3. The bar disappears. The swapping has been undone.

8.3.4 Changing the data type

Description   In the Signal Editor, the data type is displayed by an icon.

Icon	Description
±	Integer data type with sign (depending on the length: SINT, INT, LINT or DINT)
+	Integer data type without sign (depending on the length: USINT, UINT, ULINT or UDINT)
❘⬤	Digital data type (depending on the length: BYTE, WORD, DWORD or LWORD)

(The exact data type of a signal is displayed in the **I/O Mapping** window.)

The data type must changed e.g. if a signal needs to be used as an analog output or input but is only designated as a digital data type in the device description file.

Precondition   ◼ The Signal Editor is open.

Procedure   1. Click on the icon for the sign on the right-hand side of the input or output column. The icon changes.
2. Click until the desired symbol is displayed.

8.3.5 Changing signal names

Precondition   ◼ The Signal Editor is open.

Procedure   1. Click on the name of the input or output in the right-hand column. The name can now be edited.
2. Enter the desired name and confirm with the Enter key.

   The name must be unambiguous within the current view of the Signal Editor.

The changed name is displayed in the **I/O Mapping** window.

8.4 Mapping the bus I/Os

8.4.1 I/O Mapping window

Overview
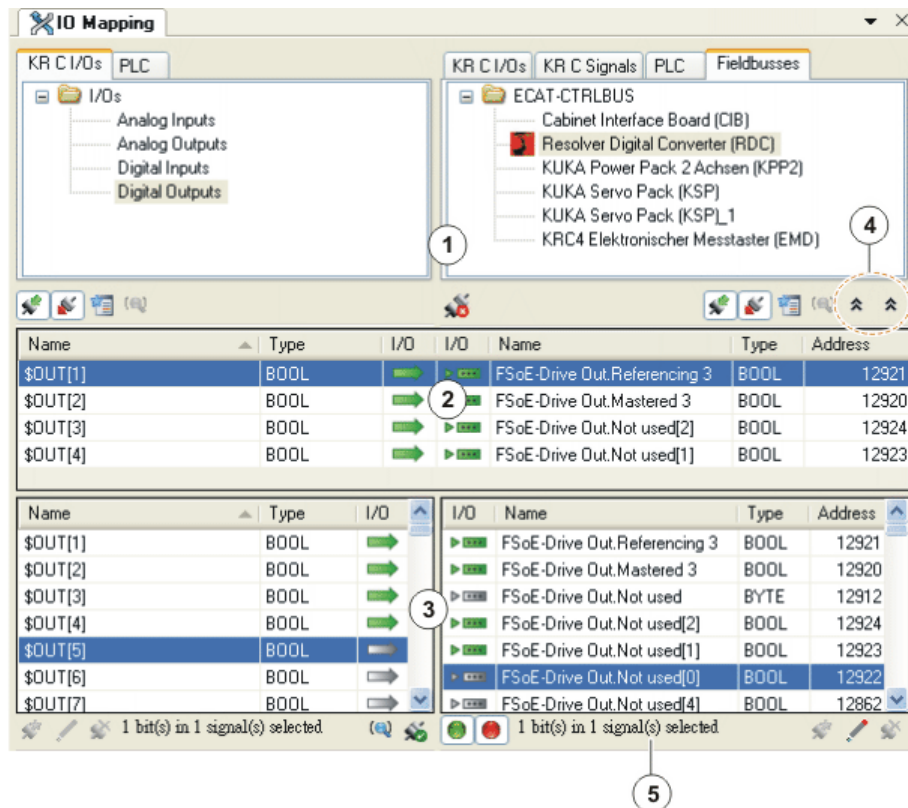
Fig. 8-4: I/O Mapping window

Item	Description
1	Display of the input/output types and field bus devices. The two areas to be connected are selected on the left and right via tabs.
	The signals in the areas selected here are displayed in the windows lower down.
2	Display of the mapped signals.
3	Display of all signals. The inputs/outputs can be mapped here.
	(>>> 8.4.3 "Mapping input to output" Page 83)
4	This allows the two signal displays to be collapsed and expanded independently of one another.
5	Displays how many bits the selected signals contain.

Mapped signals are indicated by green symbols.

Multiply mapped signals are indicated by double arrows:

(>>> 8.4.2 "Buttons in the I/O Mapping window" Page 82)

Tabs:

Name	Description
KR C I/Os	Here the analog and digital inputs/outputs of the robot controller are displayed. There is a **KR C I/Os** tab on the left and on the right. This makes it possible to map inputs and outputs of the robot controller with each other.
PLC	These tabs are only relevant if Multiprog is used.
KR C signals	Here the other signals of the robot controller are displayed.
Field buses	Here the inputs/outputs of the field bus devices are displayed. There is a **Field buses** tab on the left and on the right. Only bus inputs are displayed on the left, and only bus outputs are displayed on the right. From System Software version 8.3, these tabs can be used to map the inputs and outputs of 2 field buses.

8.4.2 Buttons in the I/O Mapping window

Filter/ Search

Some of these buttons are displayed in several places. In such cases, they refer to the side of the **I/O Mapping** window on which they are located.

With some buttons, the tool tip changes depending on whether the signals they refer to are shown or hidden.

Button	Name / description
	Filter inputs/**Display all input signals**: Shows and hides the inputs.
	Filter outputs/**Display all outputs**: Shows and hides the outputs.
	Filter dialog: The **Filter signals** window is opened. Enter filter criteria (name, data type and/or signal range) and click on the **Filter** button. Signals that conform to these criteria are displayed. When a filter is set, the button displays a green check mark at bottom right. To remove a filter that has been set, click on the button and, in the **Signals** window, click on the **Reset** button and then on **Filter**.
	Buttons above the "connected signals" window: **Search for the connected signal**: Only available if a mapped signal is selected. (>>> 8.4.7 "Searching for assigned signals" Page 86) Button below the "all signals" window: **Search text**: Displays a search box. Here it is possible to search the signal displays for signal names (or parts of names), either upwards or downwards. When the search box is displayed, the button displays a cross at bottom right. To hide the search box again, click on the button.
	Filter connected signals/**Display all connected signals** Shows and hides the connected signals.
	Filter disconnected signals/**Display all unconnected signals** Shows and hides the unconnected signals.

Mapping

Button	Name / description
	Disconnect: Disconnects the selected mapped signals. It is possible to select and disconnect a number of connections simultaneously.
	Connect: Connects signals which are selected in the "all signals" display. It is possible to select a number of signals on both sides and connect them at one go. (Only possible if the same number of signals is selected on both sides.)

Edit

Button	Name / description
	Creates signals at the provider Only relevant if Multiprog is used.
	Edit signals at the provider For field bus signals: opens an editor in which the bit assignment of the signals can be edited. (>>> 8.3 "Editing field bus signals" Page 76) Editing options are also available here for the analog inputs/outputs of the KRC and for Multiprog signals. (>>> 8.4.9 "Editing analog KRC signals" Page 87)
	Deletes signals at the provider Only relevant if Multiprog is used.

> **i** Information about Multiprog is contained in the **KUKA.PLC Multiprog** documentation.

8.4.3 Mapping input to output

Description

This procedure is used to assign the inputs and outputs of the devices to the inputs and outputs of the robot controller.

By the same principle, it is also possible to map inputs and outputs of the robot controller to each other. (In this case, the **KR C I/Os** tab must be used in both halves of the window.)

Precondition

- The robot controller has been set as the active controller.
- The bus configuration in WorkVisual corresponds to the real bus configuration.
- The field bus devices are configured.

Procedure

1. Click on the **Mapping editor** button. The **I/O Mapping** window is opened.
2. On the **KR C I/Os** tab in the left-hand half of the window, select the area of the robot controller that is to be mapped, e.g. **Digital inputs**.

   The signals are displayed in the bottom area of the **I/O Mapping** window.
3. Select the device on the **Field buses** tab in the right-hand half of the window.

   The signals of the device are displayed in the bottom area of the **I/O Mapping** window.
4. Drag the signal of the robot controller onto the input or output of the device. (Or alternatively, drag the input or output of the device onto the signal of the robot controller.)

   The signals are now mapped.

Alternative procedure for mapping:

- Select the signals to be mapped and click on the **Connect** button.
- Or: Select the signals to be mapped and select the **Connect** option from the context menu.

Multiple mapping:

It is possible to select a number of signals simultaneously (on both sides) and connect them at one go. A further possibility is as follows:

1. Select a number of signals on one side and one signal on the other side.
2. Select the option **Connect continuously** from the context menu. The signals are connected (counting upwards), starting from the one selected signal.

8.4.4 Mapping bus inputs to bus outputs by means of I/O linking (8.2)

Description   With I/O linking, a bus input can be mapped to a bus output (in the same or a different bus). This is done indirectly. A total of 3 mappings are required for this.

Procedure   1. Map the bus input to an input of the robot controller.
2. Map the robot controller input to a robot controller output.
3. Map the robot controller output to a bus output.

The input and output of the robot controller are thus multiply mapped in this case.

Schematic
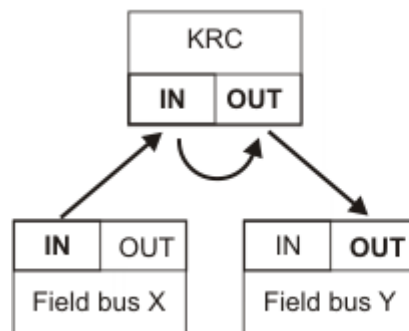
Fig. 8-5: Schematic: Mapping bus input to bus output

8.4.5 Mapping bus inputs to bus outputs using the transfer application (8.3)

Description   With the transfer application, a bus input can be mapped directly to one or more bus outputs (in the same or a different bus). For this, the **Field buses** tab is used in both halves of the window.

A maximum of 2048 bus inputs can be mapped to bus outputs. If one bus input is mapped to multiple bus outputs, the number of bus outputs is the determining factor.

If more than 512 bus inputs are to be mapped to bus outputs, the connections must be made in blocks. With block mapping, both the bus inputs and the bus outputs of a block must be adjacent, i.e. there must not be any gaps in between. Furthermore, the bus inputs and outputs of a block must be within one slot.

Example: In the PROFINET field bus, the signal names contain slots and index numbers. The index numbers stand for the bits, i.e. index number 0001 = bit 1, index number 0002 = bit 2, etc. If 2 adjacent bits within the same slot are mapped (e.g. bit 1 and bit 2 of slot 2), this results in 1 block mapping. However,

if 2 non-adjacent bits or 2 bits from different slots are mapped (e.g. bit 4 of slot 1 and bit 5 of slot 3), no block mapping results.

> **i** In System Software version 8.3 and higher, safe outputs have a yellow background. These outputs must not be mapped!

Schematic
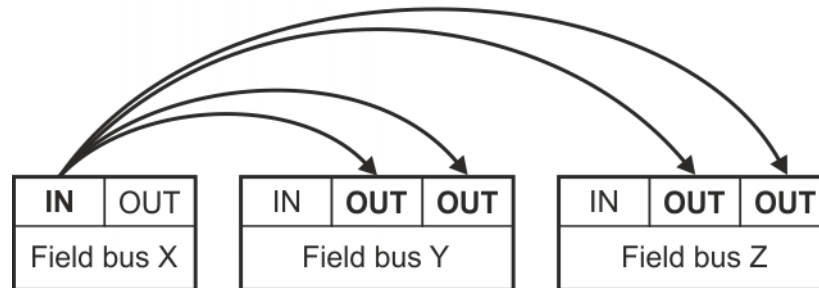
Fig. 8-6: Schematic: Mapping a bus input to bus outputs

> **i** It is possible to use a WorkVisual project with direct bus mappings with a lower System Software version. However, the mappings have no effect in the lower version.

8.4.6 Multiple mapping or reverse mapping of signals via I/O linking

Possible

Signals can be multiply mapped via I/O linking. Multiply mapped signals are indicated in the **I/O Mapping** window by a double arrow: ⬅

The following multiple mapping is possible:

- Map an input (robot controller) to an input (bus).
- Map the same input (robot controller) to one or more outputs (robot controller).
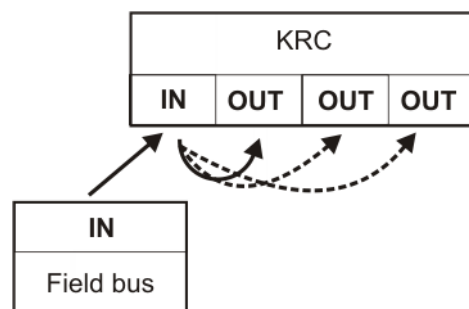
Fig. 8-7: Multiple mapping possible

The following reverse mapping is possible:

- Map an output (bus) to an input (robot controller).
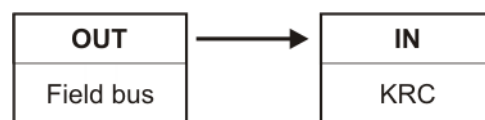
Fig. 8-8: Reverse mapping possible

Not possible

The following multiple mappings are not possible:

- Map an input (robot controller) to several inputs (bus).

■ Map an output (robot controller) to several outputs (bus).
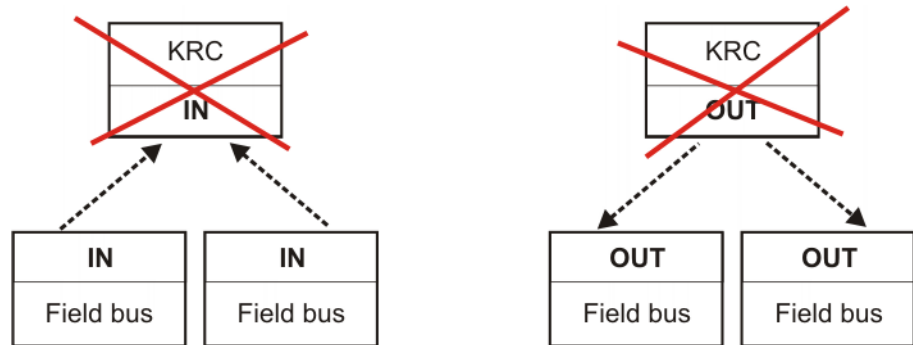
Fig. 8-9: Multiple mapping not possible

8.4.7 Searching for assigned signals

Procedure
1. Select a mapped signal.
2. In the part of the window in which the signal was selected (left-hand or right-hand half), click on the button **Searches for the connected signal**.
   ■ If a signal is mapped once: the assigned signal is now highlighted in the other half of the "all signals" window.
   ■ If a signal is multiply mapped: the **Search for signal** window opens. All signals connected with the selected signal are shown.
     Select a signal and confirm with **OK**.
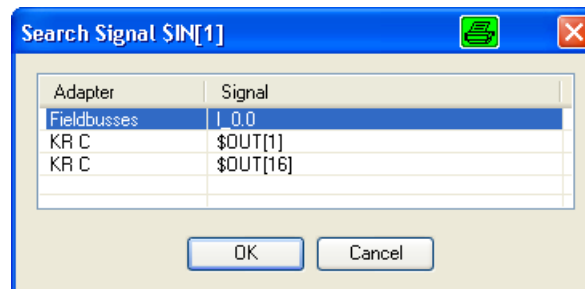
Fig. 8-10: Example: Multiply mapped signal

8.4.8 Grouping signals

Description   8 digital inputs or outputs of the robot controller can be grouped together to a signal of the data type BYTE. Grouped signals can be recognized by the name extension **#G**.

Precondition   ■ The signals that are to be grouped are not mapped.

Procedure
1. Below the **KR C I/Os** tab, select 8 consecutive signals and right-click on them.
2. Select **Group**.
   The signals are grouped to a signal of type BYTE. The name with the lowest index number is taken for the new signal.

To undo the grouping:

1. Right-click on a signal with the name extension **#G**.
2. Select **Ungroup**.

Example
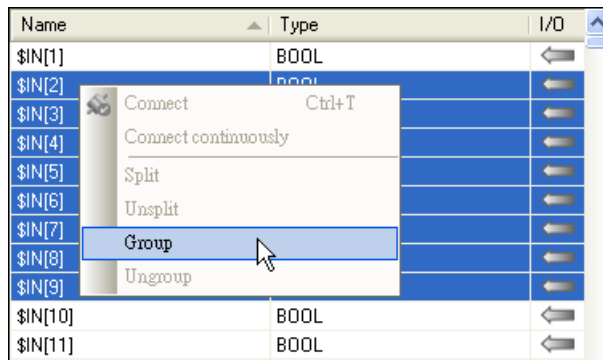
Fig. 8-11: Group signals

Fig. 8-12: A grouped signal

8.4.9 Editing analog KRC signals

Procedure

1. Select the analog signal on the left-hand **KR C I/Os** tab in the **I/O Mapping** window.

   It is also possible to select and edit several signals at once: Consecutive signals can be selected using SHIFT + click. A number of individual signals can be selected using CTRL + click.

2. At the bottom left of the **I/O Mapping** window, click on the button **Edit signals at the provider**. A window opens.

3. Enter the required calibration factor and change the data type if necessary.

4. Click on **OK** to save the data and close the window.

Box	Description
Calibration factor	The required calibration factor can be entered here.
Type	Only signals of the same type can be connected. The data type can be changed here.

8.5 Exporting the bus configuration

Description

The bus-specific configuration can be exported in the form of XML files. This export makes it possible to check the configuration files if required.

Precondition

■ The robot controller has been set as the active controller.

Procedure

1. Select the menu sequence **File** > **Import / Export**. A window opens.

2. Select **Export I/O configuration to .XML files** and click on **Next**.

3. Specify a directory. Click on **Next**.

4. Click on **Finish**.

5. The configuration is exported to the specified directory. If the configuration was successfully completed, this is indicated by a message. Close the window.

9 Long texts

9.1 Displaying / editing long texts

Precondition
- A robot controller has been added and set as active.

Procedure
1. Select the menu sequence **Editors** > **Symbol Table Editor**.
2. The long texts are sorted thematically. In the left-hand column, select which long texts are to be displayed, e.g. **Digital inputs**.
3. In the other columns, select the language or languages that are to be displayed.
4. Edit the long texts, if necessary.

Description
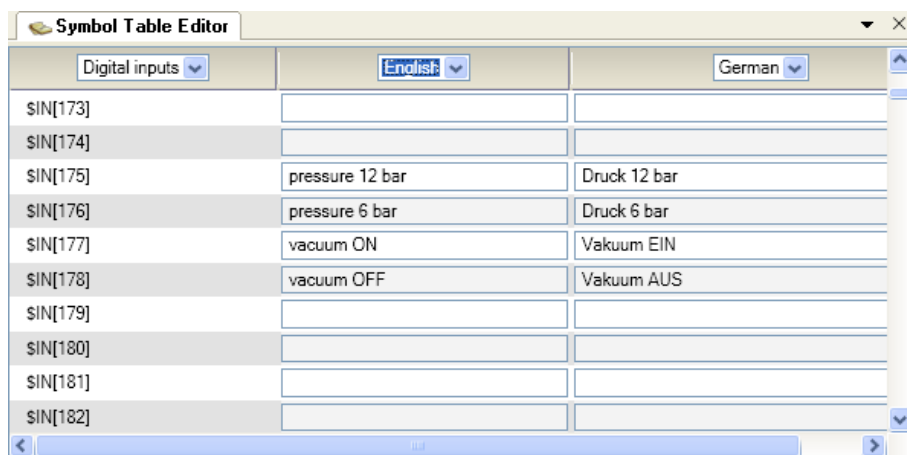
Fig. 9-1: Symbol Table Editor

> The long texts of the digital inputs/outputs can also be edited in the **I/O Mapping** window via the **Edit signals at the provider** button.

9.2 Importing long texts

Description

The following file formats can be imported:
- .TXT
- .CSV

> Imported long texts overwrite existing long texts.

Precondition
- A robot controller has been added and set as active.

Procedure
1. Select the menu sequence **File** > **Import / Export**. A window opens.
2. Select **Import Symbol Table** and click on **Next**.
3. Select the file to be imported and the language of the contained long texts.
4. If a signal already has a name and the file to be imported does not contain a name for this signal, the check box **Delete existing entries** determines what happens to the existing name.
   - **Activated**: The name is deleted.
   - **Not activated** The name is retained.
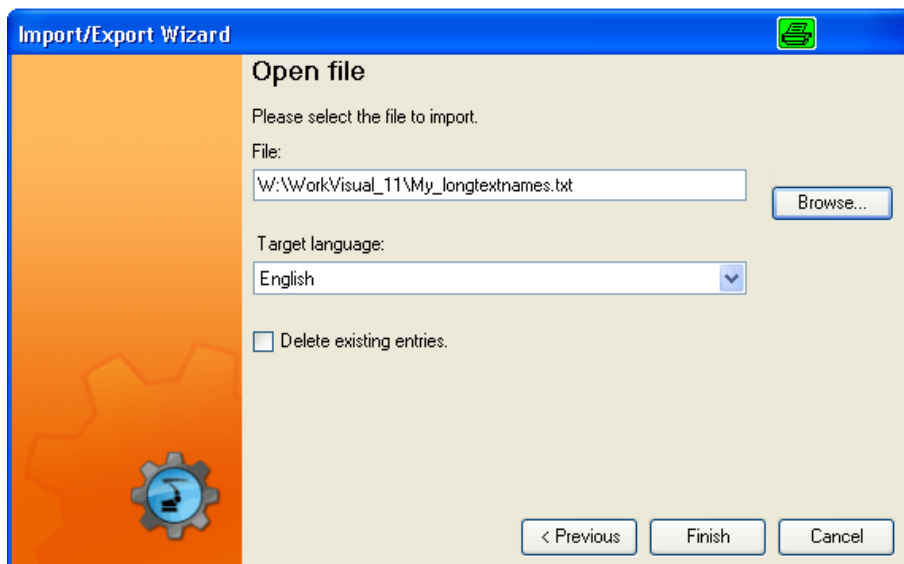
Fig. 9-2: Importing long texts

5. Click on **Finish**.

6. If the import was successful, this is indicated by a message. Close the window.

9.3 Exporting long texts

Description   Long texts can be exported in the following file formats:

- .TXT
- .CSV

Precondition   ■ A robot controller has been added and set as active.

Procedure   1. Select the menu sequence **File** > **Import / Export**. A window opens.

2. Select **Export SymbolTable** and click on **Next**.

3. Specify the path name and format of the file which is to be generated. Also select the language. Click on **Finish**.

Fig. 9-3: Exporting long texts

4. If the export was successful, this is indicated by a message in the window. Close the window.

10 Configuring KUKA buses: controller bus, system bus, extension bus

10.1 Overview

Description   The KUKA buses must be adapted in WorkVisual to the real bus configuration in the following cases:

- A device has been replaced by a device of a different type, e.g. a KPP0 by a KPP2.
- More than one device has been replaced by a device of a different type.
- One or more devices have been removed.
- One or more devices have been added.

> **i** Information about exchanging devices and about the possible device combinations is contained in the operating instructions or assembly instructions for the robot controller.

Procedure   1. Load the active project from the robot controller.

> **i** If settings have already been made in a project in WorkVisual which are to be transferred subsequently to the robot controller together with the bus configuration, then the active project must be transferred to WorkVisual using a comparison.
> (>>> 13.8 "Comparing projects (and accepting differences)" Page 145)

2. Carry out the relevant configuration work.
3. Transfer the project to the robot controller and activate it there.

10.2 Configuring the KUKA bus (8.3)

Controller Bus   Using the **Configuration proposal** functionality, the controller bus can be set up or updated automatically in WorkVisual. Insertion of individual devices, connection of devices, insertion of wagon driver files, etc., is no longer necessary.

 (>>> 6.18 "Inserting hardware components" Page 40)

The controller bus can still be edited manually if required. The procedure is the same as for the controller version 8.2.

 (>>> 10.3.1 "Inserting devices on the KUKA bus (8.2)" Page 91)

System Bus   The procedure is the same as for the controller version 8.2.
Extension Bus
 (>>> 10.3.1 "Inserting devices on the KUKA bus (8.2)" Page 91)

10.3 Configuring the KUKA bus (8.2)

10.3.1 Inserting devices on the KUKA bus (8.2)

> **i** Recommendation: Arrange the devices in the buses in the same order as they are arranged in reality.
> The arrangement has no effect on its functionality of the bus, but it is easier to edit the connections on the **Topology** tab if the arrangement already corresponds to reality.

Preparation   Only if devices are to be inserted into the Extension Bus and the node **KUKA Extension Bus (SYS-X44)** is not yet present:

1. Right-click on the node **Bus structure** on the **Hardware** tab in the **Project structure** window.
2. Select **Add** from the context menu. The **DTM selection** window is opened.
3. Select the entry **KUKA Extension Bus (SYS-X44)** and confirm with **OK**.

Precondition   ■ The device description files are present.

   ■ If devices are to be inserted into the Extension Bus, the files must be imported beforehand.

   (>>> 6.8 "Importing device description files" Page 31)

   ■ The files for the controller bus and the system bus are present in WorkVisual.

■ The robot controller has been set as the active controller.

Procedure   1. Right-click on the KUKA bus in the **Bus structure** node on the **Hardware** tab in the **Project structure** window.
2. Select **Add** from the context menu. The **DTM selection** window is opened.
3. Select the device used and confirm with **OK**. The device is inserted in the tree structure.
4. If required: Right-click on the device in the tree structure and select **Rename** from the context menu. Rename the device.

> **ℹ** In a bus, more than one device of the same type may be present. To distinguish them in the bus structure, WorkVisual automatically appends a number to the name. It is recommended, however, to give the devices meaningful names, e.g. by including the abbreviation for the installation position of the devices.
> The names that the devices have here in the bus structure are the names used in error messages.

5. Repeat steps 1 and 4 for all devices used in the real bus.
6. Check the device settings and change if necessary.

   (>>> 10.3.2 "Checking the device settings" Page 93)
7. Check the device connections and change if necessary.

   (>>> 10.3.3 "Connecting devices on the KUKA bus" Page 94)
8. Only if the change in the controller bus affects a KPP, or if the controller bus has been complete reconfigured: Insert the wagon driver configuration.

   (>>> 10.3.5 "Inserting the wagon driver configuration" Page 96)
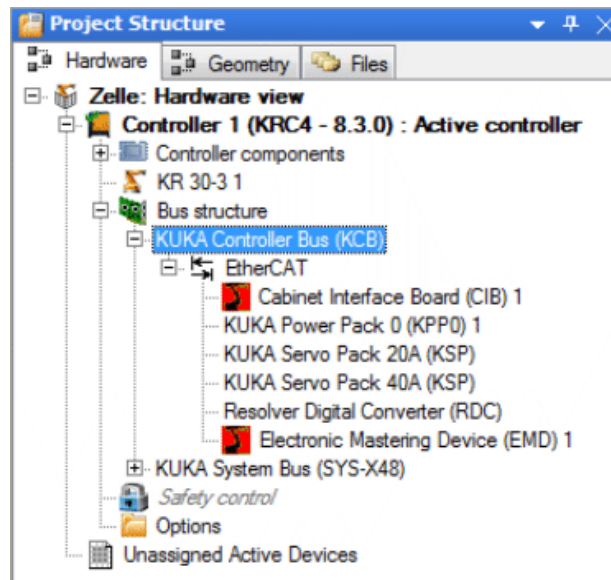
Fig. 10-1: Example of a controller bus

10.3.2 Checking the device settings

Procedure

1. Right-click on the device on the **Hardware** tab in the **Project structure** window.
2. Select **Settings…** from the context menu. The **Settings …** window opens.
3. Select the **General** tab.
4. Check that the following settings have been made. If not, correct the settings.

    - **Check vendor ID**: active
    - **Check product code**: active
    - **Check revision number**: OFF
    - **Check serial number**: inactive

5. Close the window by clicking on **OK**.
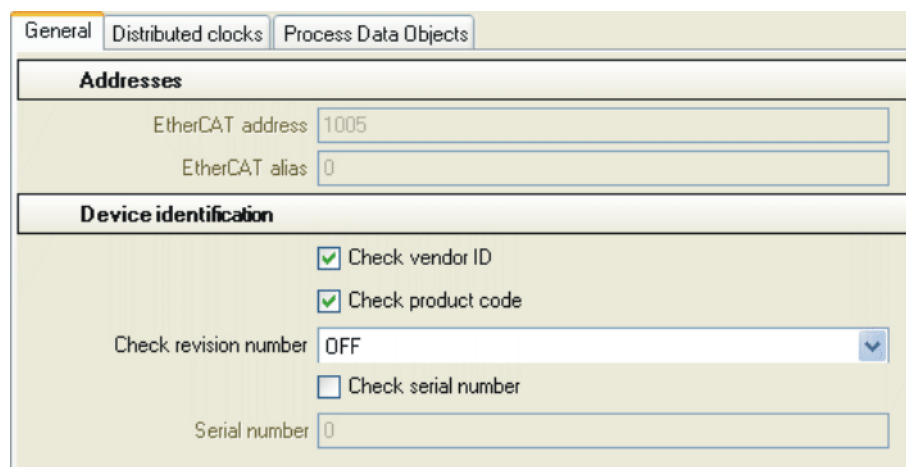
Fig. 10-2: Device settings, "General" tab

10.3.3 Connecting devices on the KUKA bus

Description   If devices are inserted into the bus, WorkVisual connects the devices automatically. As WorkVisual does not know the real bus configuration, the connections must be checked and changed where necessary.

> **i** Information about the connector pin allocation of the devices can be found in the operating instructions for the robot controller.

Procedure   1. Right-click on the bus on the **Hardware** tab in the **Project structure** window.
2. Select **Settings…** from the context menu. The **Settings…** window is opened.
3. Select the **Topology** tab.

   (>>> 10.3.4 ""Topology" tab" Page 95)
4. Select and delete invalid connections. To do this, either press the Del key, or right-click and select **Delete**.
5. Insert missing connections. To do this, click on a connection and hold down the mouse button. Drag the mouse pointer to another connection and release the mouse button.
6. Mark temporary connections as such. To do this, right-click on the connection and select **Disconnectable** from the context menu. The connection is displayed as a broken line.

   A temporary connection is e.g. with the controller bus, the connection to the Electronic Mastering Device (EMD), because the EMD is not permanently connected.
7. Click on devices for which the address or alias address is not yet correct. A window appears. Enter the correct address.

   All temporarily connected devices require an alias address. For the EMD, the alias address 2001 must be entered!
8. If required: Rearrange the devices using Drag&Drop. This serves to increase the clarity of the **Topology** tab. It has no effect on the bus.
9. Click on **OK** at bottom right.

10.3.4 "Topology" tab
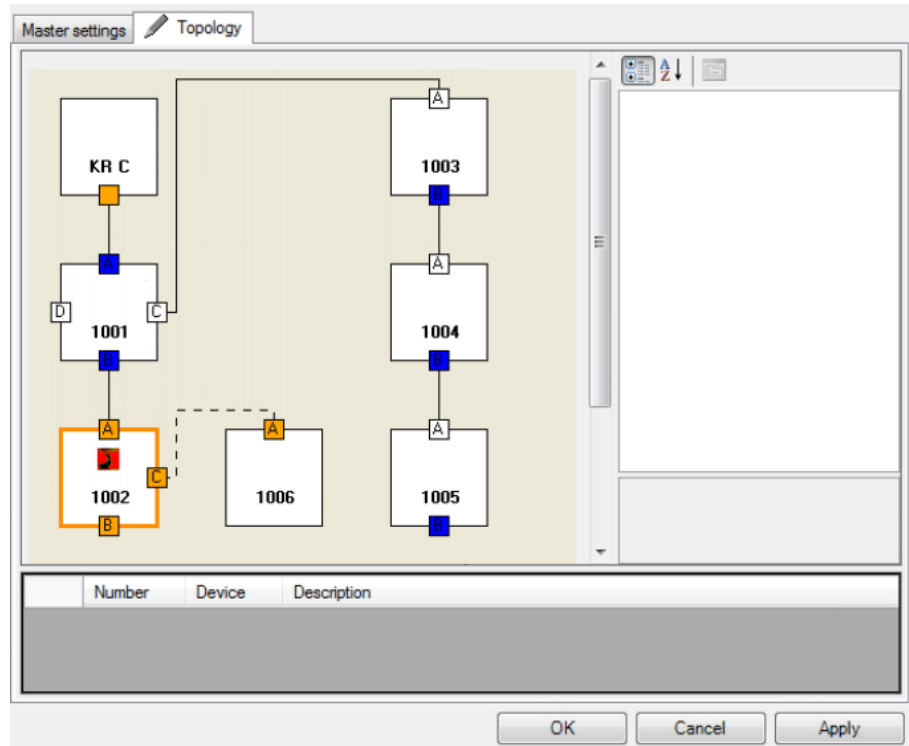
Fig. 10-3: "Topology" tab – example of a controller bus

Properties of the "Topology" tab:

■ Each device in the bus is represented by 1 rectangle.

■ The numbers of the devices specify their physical address.

■ **To display the name of a device:**

Move the mouse pointer onto the device. A tool tip is displayed with the name of the device.

Or alternatively, select the device. The window on the right displays the properties of this device, e.g. the name.

The devices in the figure:

Device	Name
1001	Cabinet Interface Board (CIB)
1002	Resolver Digital Converter (RDC)
1003	KUKA Power Pack 2 Axes (KPP2) (G1)
1004	KUKA Servo Pack Wrist Axes (KSP) (T1)
1005	KUKA Servo Pack Main Axes (KSP) (T2)
1006	Electronic Mastering Device (EMD)

■ **To display the name of a connection:**

Move the mouse pointer onto the connection. A tool tip is displayed with the name of the connection.

■ Lines shows the connections between the devices.

Solid lines indicate permanent connections. Broken lines indicate temporary connections.

■ The devices can be reordered using Drag&Drop. This serves to increase the clarity of the **Topology** tab. It has no effect on the bus.

■ The window on the right displays the properties of the selected device, e.g. the address and alias address. Some of the properties can be changed.

All temporarily connected devices require an alias address. For the EMD, the alias address 2001 must be entered!

■ The message area below the graphic shows if a device has an invalid address or alias address.

Editing connections:

■ Select and delete invalid connections.

To do this, either press the Del key, or right-click and select **Delete**.

■ Insert missing connections.

To do this, click on a connection and hold down the mouse button. Drag the mouse pointer to another connection and release the mouse button.

■ Mark temporary connections as such.

To do this, right-click on the connection and select **Disconnectable** from the context menu. A temporary connection is e.g. the connection to the Electronic Mastering Device (EMD), because the EMD is not permanently connected.

10.3.5 Inserting the wagon driver configuration

Description

The wagon driver configuration must be inserted into the WorkVisual project in the following cases:

■ If the controller bus has been completely reconfigured.

■ Or if a change has been made to the controller bus which affects a KPP.

For this, the following configuration files are required:

■ CFCoreWaggonDriverConfig.xml

■ EAWaggonDriverConfig.xml

These files are automatically installed together with WorkVisual. They are located in the directory: C:\Program Files (x86)\KUKA\WorkVisual[…]\WaggonDriverConfigurations. They can be found there in the subdirectory for the relevant controller bus variant.

Which wagon driver configuration should be used for which controller bus?

Controller bus with …	Directory
KPP without external axis	**KPP 600-20**
KPP with 1 external axis	**KPP 600-20-1x40 (1x64)**
KPP with 2 external axes	**KPP 600-20-2x40**
KPP with 1 external axis and ServoGun FC Sensor Box	**KPP 600-20-1x40 + SDC**
KPP with 2 external axes and ServoGun FC Sensor Box	**KPP 600-20-2x40 + SDC**
4-axis palletizer with external axis	**4Ax_PA_mit_ZA**
4-axis palletizer without external axis	**4Ax_PA_ohne_ZA**
5-axis palletizer with external axis	**5Ax_PA_mit_ZA**
5-axis palletizer without external axis	**5Ax_PA_ohne_ZA**
AGILUS sixx	**6Ax_CIBsr_KPPsr_KSPsr**

Precondition

■ The robot controller has been set as the active controller.

Procedure

1. Expand the node of the robot controller on the **Files** tab in the **Project structure** window.

2. Then expand the following subordinate nodes: **Config** > **User** > **Common** > **Mada**.

3. Only if there are already wagon driver files in the **Mada** directory and these need to be removed:

    ▪   Right-click on a file and select **Delete** from the context menu.
    ▪   Repeat for the second file.

4. Right-click on the **Mada** folder and select **Add external file** from the context menu.

5. A window opens. In the **File type** box, select the entry **All files (*.*)**.

6. Navigate to the directory in which the files for the wagon driver configuration are located, select the files and confirm with **Open**.

    The files are now displayed in the tree structure below the **Mada** folder. (If not: collapse and expand all the folders in order to refresh the display.)

10.4   Assigning FSoE slave addresses (≥ 8.2.21 and 8.3)

> **i**   This description is valid for the controller variant (V)KR C4. For KR C4 compact, please contact KUKA.
> (>>> 15 "KUKA Service" Page 173)

Description

The robot controller is supplied with preconfigured FSoE addresses. The user needs to assign the addresses via WorkVisual only in the following cases:

▪   More than 1 RDC is connected.

    On delivery, RDCs are preconfigured with the FSoE address "2". An address must not be used more than once on a robot controller.

▪   Several devices of the same type have been replaced at the same time.

> **i**   The assignment of addresses to devices is predefined.
> (>>> 10.4.1 "FSoE addresses" Page 99)

Preparation

▪   Determine the KUKA serial number of the real device.

    ▪   With KSPs and KPPs, the serial number is given on the rating plate.

        There may be several serial numbers on the Lenze modules. The KUKA number is the relevant one.

        (>>> 10.4.2 "Determining the serial number for the Lenze KSP/KPP" Page 100)

    ▪   RDCs have a label with a barcode on the board. The serial number is specified in encrypted form.

        (>>> 10.4.3 "Determining the serial number of the RDC" Page 100)

▪   Determine the IP address of the interface via which the WorkVisual PC is connected to the robot controller:

    a. On the robot controller, select **Diagnosis** > **Diagnostic monitor** from the main menu.

    b. Select the interface in the **Module** box: **Network interface (Service)** (for KSI) or **Network interface (KLI)**.

        The relevant data for the interface, including the IP address, are now displayed.

Precondition

WorkVisual:

▪   The IP address of the WorkVisual PC is in the same subnet as the IP address of the interface to which it is connected (KSI or KLI).

> ℹ️ If DHCP is used, a suitable IP address is assigned automatically.
> If a fixed IP address is used in conjunction with a connection via the KLI, the address depends on the interface setting that is configured as a Windows interface in the KLI. (Default: "virtual5" with the IP address 172.31.1.147 and the subnet mask 255.255.0.0)
> If a fixed IP address is used in conjunction with a connection via the KSI, the area of the suitable IP address depends on the settings of the non-real-time communication. (Default: IP address 172.17.x.x with subnet mask 255.255.0.0)

- Network connection to the real robot controller
- The configuration in WorkVisual is identical to the real bus architecture.

  This can be achieved most reliably by loading the active project of the real robot controller in WorkVisual.

- The robot controller has been set as the active controller.
- The affected devices support address assignment by software.

  This is defined in the device description file. In certain cases, this feature may be present in the latest device description file, although the device was inserted into the bus structure in WorkVisual with an older version of the file. The new file must then be imported in WorkVisual and the device must be removed from the bus structure and then reinserted.

Real robot controller:

- User group Expert or higher
- Operating mode T1
- The safety controller does not give a drives enable signal.
  - This status can be brought about in the following way: trigger an EMERGENCY STOP.
  - This status can be checked as follows: In the status bar, click on the **Drives** status indicator. The **Motion conditions** window opens. The box **Drives enabled from Safety** must be gray. It must not be green.

> ℹ️ Information about the status bar can be found in the operating and programming instructions for the system software.

Procedure

1. Double-click on the **KUKA Controller Bus (KCB)** node on the **Hardware** tab in the **Project structure** window. The **Settings …** window opens.
2. Enter the IP address (KSI or KLI) determined during the preparation stage. Click on **OK** to save the entries and close the window.
3. Right-click on the **KUKA Controller Bus (KCB)** node and select **Connect** from the context menu.

   The node is now displayed in green italics.
4. Right-click on the relevant device under the **KUKA Controller Bus (KCB)** node and select **Connect** from the context menu.

   The device description is now displayed in green italics.
5. Right-click again on the device and select **Functions** > **Assign FSoE slave address...** from the context menu. The **FSoE slave address assignment** window opens and the currently set FSoE address is displayed.
6. Enter the serial number and the new FSoE address. Leading zeros may be omitted.

   WorkVisual recognizes whether the serial number is correct: If it is not, a red exclamation mark is displayed to the left of the box. This is also the case during entry, while the number is still incomplete and thus incorrect.

   The red exclamation point disappears as soon as the number is complete and correct.

7. If the serial number is correct, click on **Accept**. Then click on **OK**. The window closes.

8. Right-click again on the device and select **Disconnect** from the context menu.

    The data are now saved on the real device. The real controller bus cannot yet access the device, however.

9. Right-click on the **KUKA Controller Bus (KCB)** node and select **Disconnect** from the context menu.

10. Reconfigure the I/O drivers on the real robot controller.

    Once the reconfiguration is complete, the controller bus once again has access to its devices and to the current addresses.

10.4.1 FSoE addresses

KSP and KPP

Controller variant with 1 row of drive modules		
KSP, left Address: **1022**	KSP, middle Address: **1021**	KPP, right Address: **1020**

Controller variant with 2 rows of drive modules		
KSP, top left Address: **1032**	KSP, top center Address: **1031**	KPP, top right Address: **1030**
KSP, bottom left Address: **1022**	KSP, bottom center Address: **1021**	KPP, bottom right Address: **1020**

RDC The following tables display common topologies and how the FSoE addresses must be assigned to them.

- **RDC-Rob** = RDC of a KUKA robot (except KR 1000 titan)
- **RDC-Titan** = RDC of a KR 1000 titan
- **RDC-ExAx** = RDC for external axes or CKs

The KR 1000 titan has 2 RDCs. These must always have the FSoE addresses "2" and "3".

X21, cascaded	X21	X21.1, cascaded	X21.1
RDC-Rob Address: **2**	---	--- **RDC-ExAx** Address: **3** **RDC-ExAx** Address: **3**	--- --- **RDC-ExAx** Address: **4**

X21, cascaded	X21	X21.1, cascaded	X21.1
RDC-Titan Address: **2**	**RDC-Titan** Address: **3**	--- **RDC-ExAx** Address: **4** **RDC-ExAx** Address: **4**	--- --- **RDC-ExAx** Address: **5**

X21, cascaded	X21	X21.1, cascaded	X21.1
RDC-ExAx Address: **2**	**RDC-Rob** Address: **3**	---	---
		RDC-ExAx Address: **4**	---
		RDC-ExAx Address: **4**	**RDC-ExAx** Address: **5**

Additional components

It is possible to assign FSoE addresses to the following components. In practice, however, it is not necessary, as they only occur once per robot controller.

Component	Address
smartPAD	**13330**
SIB	**13331**
Extended SIB	**13332**
CIB	**1**

> The addresses for smartPAD, SIB, Extended SIB and CIB must never be assigned to other components.

10.4.2 Determining the serial number for the Lenze KSP/KPP

Fig. 10-4: Example: Labels on the Lenze KSP/KPP

Item	Description
1	**KUKA Ser.-Nr.**: This is the the relevant number.
2	**Ser.-Nr.**: This number is not relevant.

10.4.3 Determining the serial number of the RDC

Description

RDCs have a label with a barcode on the board. The serial number is specified in encrypted form. There are 2 types of barcode. The length of the serial number depends on the type.

If the RDC is located in the RDC box, the box must be opened in order to read the label.

> Information about the RDC and opening the RDC box can be found in the operating instructions for the robot controller.
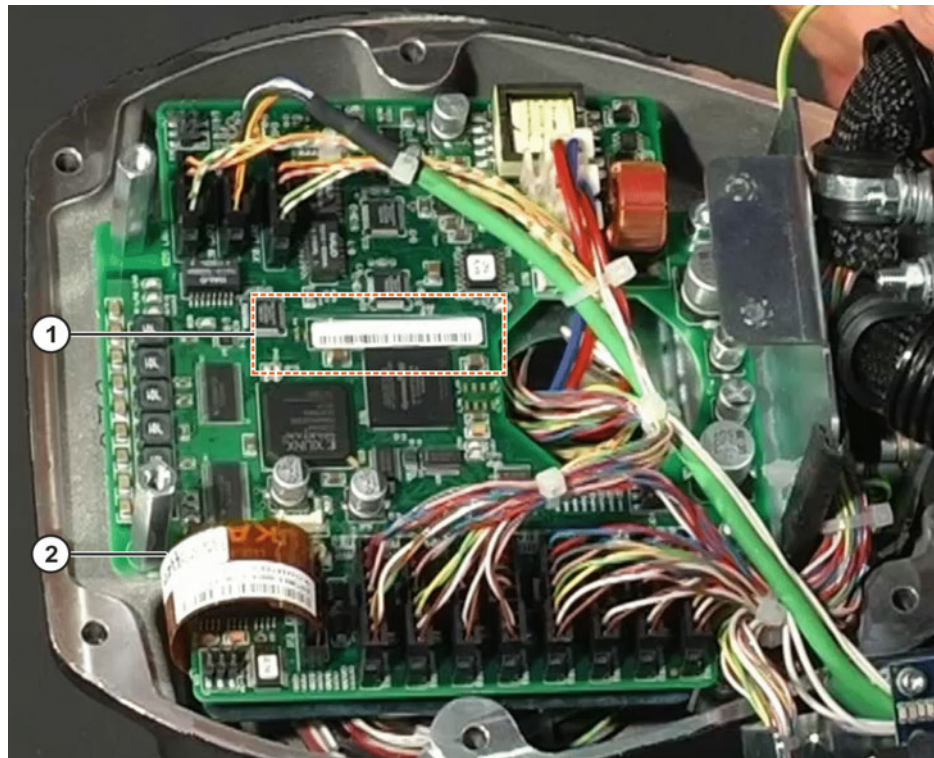
Label on the RDC

Fig. 10-5: Example: open box with RDC

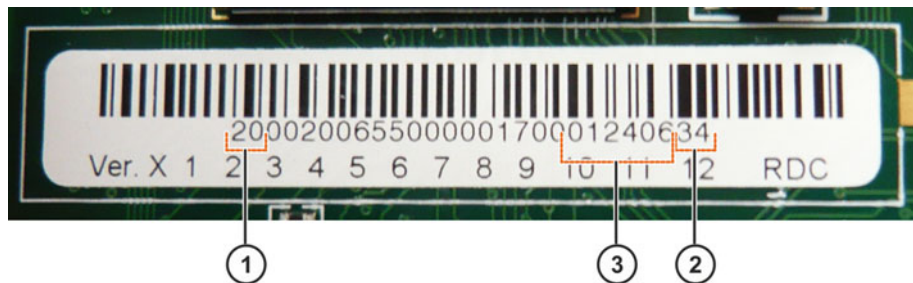| Item	Description
1	The relevant label is located in the center of the board.
2	The labels on the EDS memory strip are not relevant.

Encrypted serial number

Fig. 10-6: Example: label on an RDC

Item	Description
1	The two leftmost digits specify the type. Possible values are: ■ Type 20 (in the example) ■ Type 26
2	The two rightmost digits specify the check digit. The check digit itself is not relevant.
3	To the left of the check digit is the serial number. The number of figures it comprises is dependent on the type: ■ With type 20: 6 digits (in the example: 012406) ■ With type 26: 7 digits

10.5 Assigning FSoE slave addresses (≤ 8.2.20)

> **i** This description is valid for the controller variant (V)KR C4. For KR C4 compact, please contact KUKA.
> (>>> 15 "KUKA Service" Page 173)

Description

The robot controller is supplied with preconfigured FSoE addresses. Only when a device is replaced must the user assign it an address in WorkVisual.

It is not possible to replace more than one device of the same type at the same time.

> **i** The assignment of addresses to devices is predefined.
> (>>> 10.4.1 "FSoE addresses" Page 99)

Preparation

- Determine the KUKA serial number of the real device.
  - With KSPs and KPPs, the serial number is given on the rating plate.
    There may be several serial numbers on the Lenze modules. The KUKA number is the relevant one.
    (>>> 10.4.2 "Determining the serial number for the Lenze KSP/KPP" Page 100)
  - RDCs have a label with a barcode on the board. The serial number is specified in encrypted form.
    (>>> 10.4.3 "Determining the serial number of the RDC" Page 100)
- Determine the IP address of the interface via which the WorkVisual PC is connected to the robot controller:
  a. On the robot controller, select **Diagnosis** > **Diagnostic monitor** from the main menu.
  b. In the **Module** box, select the entry **Networkinterface (KLI)**.
    The relevant data for the interface, including the IP address, are now displayed.

Precondition

WorkVisual:

- The IP address of the WorkVisual PC is in the same subnet as the IP address of the interface to which it is connected (KLI).
- Network connection to the real robot controller
- The configuration in WorkVisual is identical to the real bus architecture.
  This can be achieved most reliably by loading the active project of the real robot controller in WorkVisual.
- The robot controller has been set as the active controller.
- The affected devices support address assignment by software.
  This is defined in the device description file. In certain cases, this feature may be present in the latest device description file, although the device was inserted into the bus structure in WorkVisual with an older version of the file. The new file must then be imported in WorkVisual and the device must be removed from the bus structure and then reinserted.

Real robot controller:

- User group Expert or higher
- Operating mode T1
- $USER_SAF == TRUE

$USER_SAF == TRUE

The conditions under which $USER_SAF is TRUE depend on the controller variant and the operating mode:

Controller	Operating mode	Condition
KR C4	T1, T2	■ The enabling switch is pressed.
	AUT, AUT EXT	■ The physical safeguard is closed.
VKR C4	T1	■ The enabling switch is pressed. ■ E2 is closed.
	T2	■ The enabling switch is pressed. ■ E2 and E7 are closed.
	AUT EXT	■ The physical safeguard is closed. ■ E2 and E7 are open.

Procedure

1. Double-click on the **KUKA Controller Bus (KCB)** node on the **Hardware** tab in the **Project structure** window. The **Settings …** window opens.

2. Enter the IP address of the KLI determined during the preparation stage. Click on **OK** to save the entries and close the window.

3. Right-click on the **KUKA Controller Bus (KCB)** node and select **Connect** from the context menu.

   The node is now displayed in green italics.

4. Right-click on the relevant device under the **KUKA Controller Bus (KCB)** node and select **Connect** from the context menu.

   The device description is now displayed in green italics.

5. Right-click again on the device and select **Functions** > **Assign FSoE slave address...** from the context menu. The **FSoE slave address as-signment** window opens and the currently set FSoE is displayed.

6. Enter the serial number and the new FSoE address. Leading zeros may be omitted.

   WorkVisual recognizes whether the serial number is correct: If it is not, a red exclamation mark is displayed to the left of the box. This is also the case during entry, while the number is still incomplete and thus incorrect.

   The red exclamation point disappears as soon as the number is complete and correct.

7. If the serial number is correct, click on **Accept**. Then click on **OK**. The win-dow closes.

8. Right-click again on the device and select **Disconnect** from the context menu.

   The data are now saved on the real device. The real controller bus cannot yet access the device, however.

9. Right-click on the **KUKA Controller Bus (KCB)** node and select **Discon-nect** from the context menu.

10. Select **Shutdown** from the main menu on the robot controller and select the options **Cold start** and **Reload files**.

    The controller is rebooted. The controller bus now has access to its devic-es and to the current addresses again.

11 RoboTeam

> ℹ This documentation describes how RoboTeams can be configured offline with WorkVisual. Basic information about RobotTeam and about installation and programming on the robot controller can be found in the documentation **KUKA.RoboTeam**.

> ℹ The "RoboTeam" functionality is not available for projects with VKR C4 robot controllers.

11.1 Creating a RoboTeam

11.1.1 Creating a new RoboTeam project

Description   WorkVisual provides templates which enable creation of a new project containing one or more RoboTeams. A wizard, the **Cell Configuration Wizard**, guides the user through the creation process.

Templates   Templates for projects containing RoboTeams:

| Template	Description
Generic RoboTeam project	Creates a project in which the number of RoboTeams and independent robots is specified by the user. The user also specifies the number of robots and external axes in each RoboTeam.
Simple RoboTeam project	Creates a project with 1 RoboTeam. The RoboTeam contains 2 robots and 1 external axis.
Project with two RoboTeams	Creates a project with 2 RoboTeams. Each RoboTeam contains 2 robots and 1 external axis. The project also contains a handling robot.
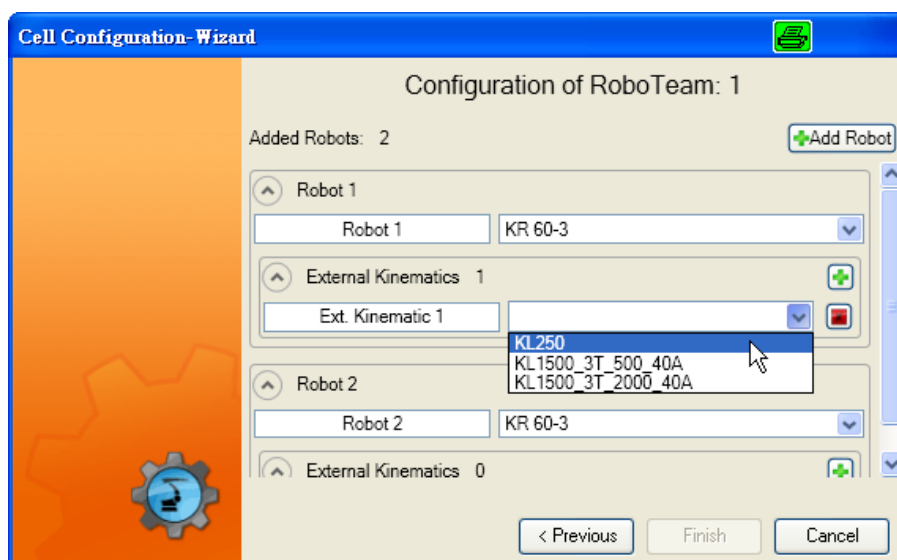
Fig. 11-1: Example screen: Cell Configuration Wizard

Procedure   1. Click on the **New…** button. The **Project Explorer** is opened. On the left, the **Create
project** tab is selected.
2. Select one of the templates for a RoboTeam project in the **Available templates** area of the tab.

3. Enter a name for the project in the **File name** box.

4. The default directory for projects is given in the **Location** box. If required, select a different directory.

5. Click on the **New** button. The Cell Configuration Wizard opens.

6. Make the required settings in the wizard, e.g. select the robot type. Clicking on **Next** takes you to the next screen.

7. Once all the relevant settings have been made, click on **Finish** and then, in the next screen, on **Close**.

8. The robot network is now displayed on the **Hardware** tab of the **Project structure** window, and also in the **Cell configuration** window. The robot network is displayed in a tree structure on the **Hardware** tab and graphically on the **Cell configuration** tab.
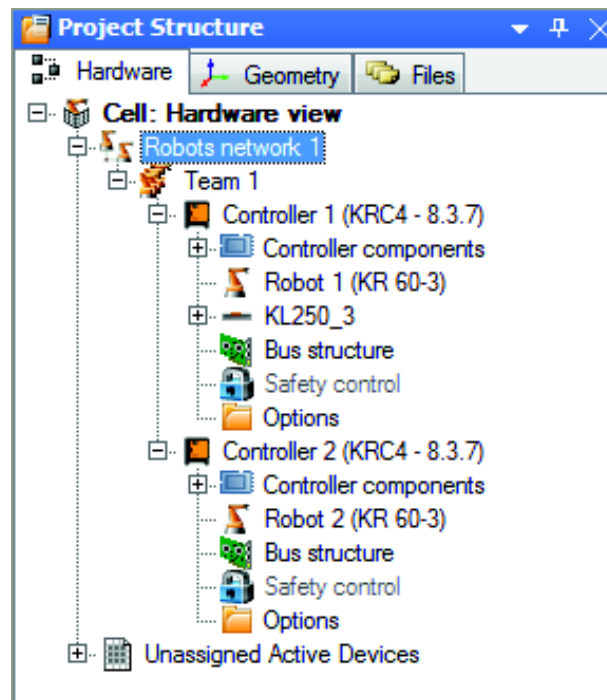
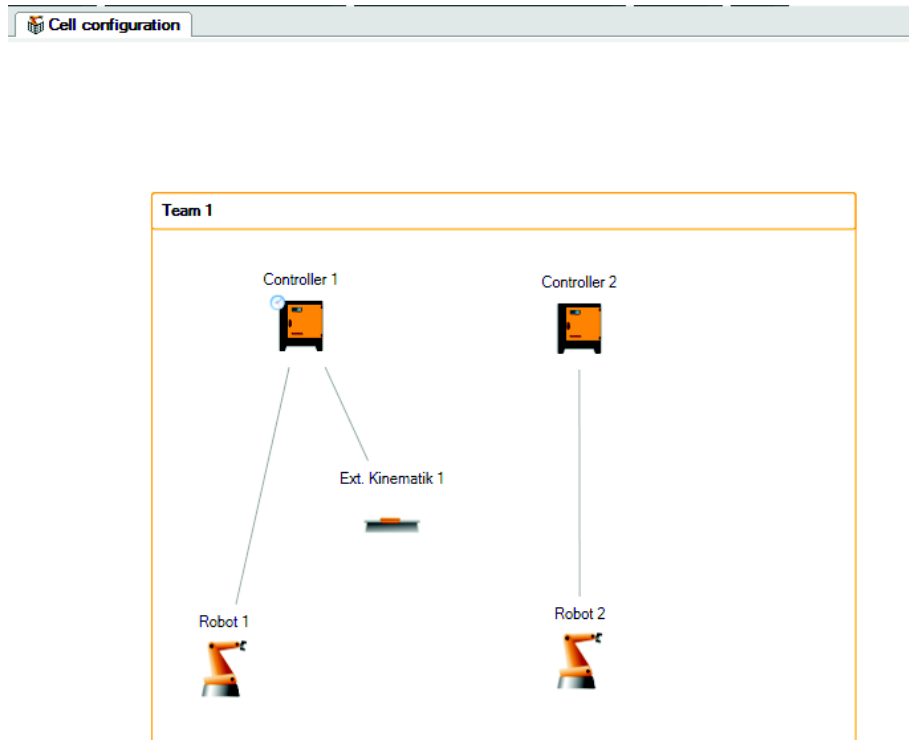**Fig. 11-2: Example: RoboTeam on the Hardware tab**

Fig. 11-3: Example: RoboTeam in the Cell configuration window

11.1.2 Inserting a RoboTeam in an existing project

Description

A RoboTeam can be inserted in an existing project. This can be done via the **Project structure** or **Cell configuration** window.

> If the RoboTeam is to be inserted in a new project, specific templates are available for project creation. This is quicker and easier than first creating an ordinary project and then inserting the RoboTeam.
> (>>> 11.1.1 "Creating a new RoboTeam project" Page 105)

Precondition

■ The catalogs **KukaControllers** and **KUKARobots[…]** are available.
■ If the RoboTeam is to contain external axes: The catalog **KukaExternalKinematics[…]** is available.

Procedure: Project structure

1. Right-click on the node of the cell on the **Hardware** tab in the **Project structure** window and select the option **Add RoboTeam** from the context menu.

   The node **Robot network** and the subnode **RoboTeam** are inserted. The nodes are numbered by default. They can be renamed.
2. Add the required number of robot controllers to the **RoboTeam** node.
3. Assign robots to the robot controllers.
4. If required, assign external axes to the robot controllers.
5. If required, a further RoboTeam can be added to the network.

   To do this, right-click on the **Robot network** node and select the option **Add RoboTeam** from the context menu. Then repeat steps 2 to 5.

Procedure: Cell configuration

1. Right-click in the empty space in the **Cell configuration** window and select the option **Add RoboTeam** from the context menu.

   A new RoboTeam is added. The name can be changed.
2. Add the required number of robot controllers to the new RoboTeam.

3. Assign robots to the robot controllers.

4. If required, assign external axes to the robot controllers.

5. If required, a further RoboTeam can be added to the network.

   Right-click in the empty space and select the option **Add RoboTeam** from the context menu. Then repeat steps 2 to 5.

11.2 Configuring the RoboTeam

11.2.1 Defining the time master

Description

Once a robot network has been created, the first robot controller in the tree structure is defined by default as the time master by WorkVisual. This assignment can be changed.

In the **Cell configuration** window, the time master is indicated by a small analog clock icon. There can be only 1 time master per network.

On the real robot controller, the time master is not visible and cannot be changed.

Precondition

■ The **Cell configuration** window is open.

Procedure

1. Right-click on the robot controller that is to be set as the new time master.

2. Select **RoboTeam** > **Set as time master** in the context menu.

   The new time master is now indicated by the analog clock icon. The clock has now disappeared from the previous time master.
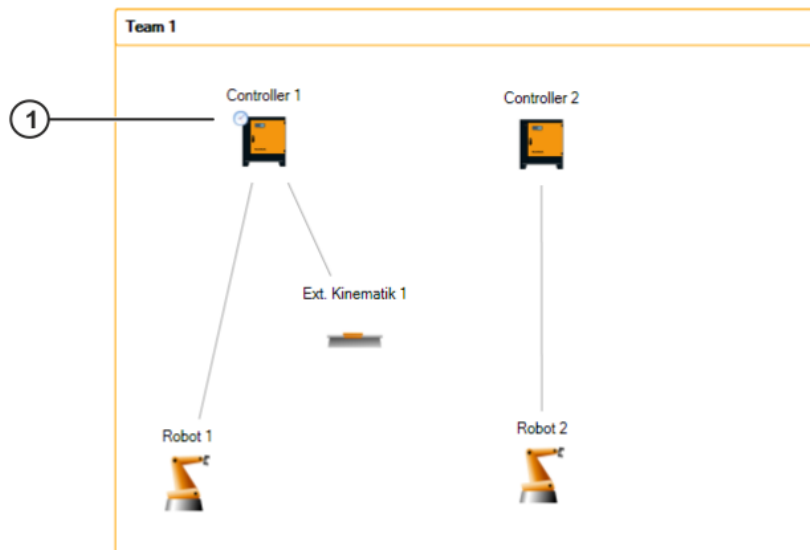

Fig. 11-4: Display of the time master

Item	Description
1	The clock indicates the time master.

11.2.2 Defining the motion master

Description   This procedure is used to define which kinematic system should be able to follow the motions of another kinematic system. The second kinematic system is then the motion master. This procedure therefore does not define which kinematic systems actually follow the motions of others, but merely which possibilities should apply. Only those kinematic systems that have been defined here in WorkVisual as master and/or slave can be used as such in the program.

Multiple connections and bidirectional connections are possible, as kinematic systems may be motion master and/or slave during program execution.

When a connection is created, the required BASE and TOOL coordinate systems are automatically created. These are identified by the names **REF_PIN** and **LK_BASE** and an icon. In the case of a robot that is to follow another robot, a reference to the TOOL and BASE coordinate systems of the other robot is created. The coordinate systems are marked in red in the **Tool/base management** editor, as they are not yet assigned to any number.

> **i** It is advisable to create only those connections which are actually required in the program. Do not connect every kinematic system to every other kinematic system in both directions, as this would take up TOOL and BASE coordinate systems unnecessarily. These would then be unavailable for other purposes.

Precondition   ■ The **Cell configuration** window is open.

Procedure   1. Click on a kinematic system (robot or external axis) and hold down the mouse button.

2. Drag the mouse pointer to another kinematic system and release the mouse button. The pop-up window **{Kinematic1} should follow {Kinematic2}** opens.

3. Click on the pop-up window.

   The first kinematic system can now follow the other kinematic system. This is indicated by a dotted arrow. The arrow is red if the root points have not yet been calibrated. A green arrow indicates that the root points have already been calibrated.

4. Repeat steps 1 to 3 until every kinematic system is connected to at least one other kinematic system.

5. Open the **Tool/base management** editor and assign numbers as desired to the BASE and TOOL coordinate systems that have not yet been assigned.
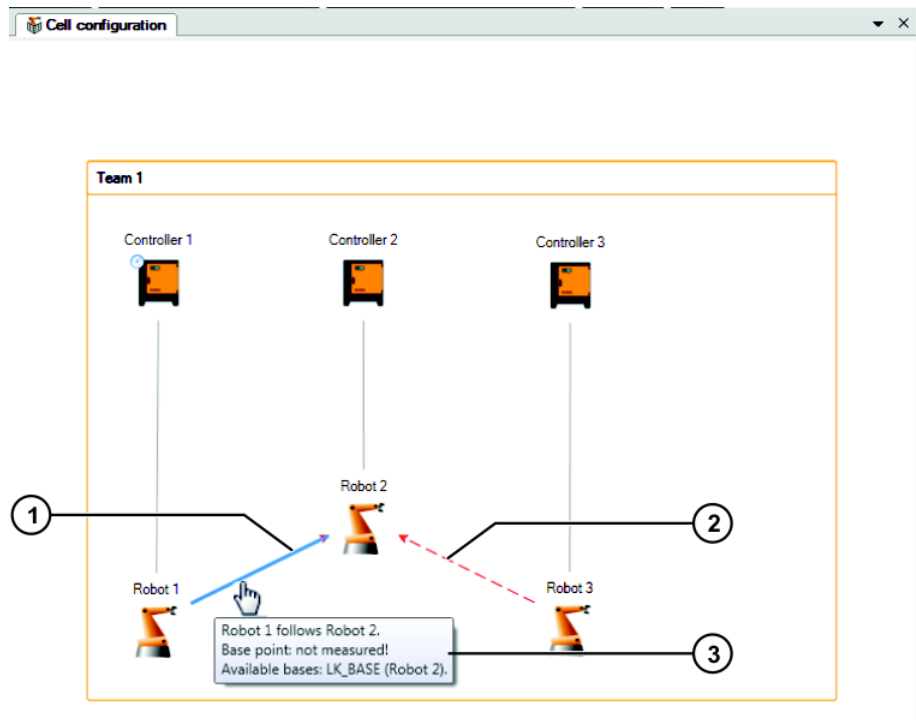
Fig. 11-5: Display of the motion master

Item	Description
1	Robot 2 is the motion master of Robot 1. The arrow is blue instead of red because the mouse pointer is positioned on it.
2	Robot 2 is the motion master of Robot 3.
3	Positioning the mouse pointer on an arrow opens an info box.

Fig. 11-6: TOOL and BASE coordinate systems in the RoboTeam

Item	Description
1	TOOL coordinate system that is not assigned to any number
2	Indicates that the coordinate system belongs to a connection of two robots
3	BASE coordinate system that is not assigned to any number

11.2.3 Deleting a master/slave link

Description   Links which define motion masters can be deleted if they are no longer needed.

Procedure
1. Click on the arrow to be deleted. The color of the arrow changes to blue.
2. Right-click and select **Delete** from the context menu. The arrow is deleted.

11.2.4 Creating and configuring semaphores

Description   The semaphore is used to defined the order in which the robots are allowed to enter the workspace.

> **i** In the KUKA.RoboTeam technology package, the semaphores are labeled as workspaces. These workspaces are specific to RoboTeam and have nothing to do with the geometric workspaces derived from the individual axis ranges of the robot.
> If KUKA.RoboTeam is installed on the robot controller, the semaphores can be displayed on the KUKA smartHMI via **Display** > **Workspace**.

Precondition
- The RoboTeam has been created.
- The **Cell configuration** window is open.

Procedure
1. Right-click on a robot controller and select **RoboTeam** > **Create semaphore** from the context menu.
2. A semaphore is inserted. The default name is **Semaphore** *[consecutive no.]*. The default name can be changed.
3. The robot controller selected on creation has the priority 1. The priority can be changed:
   a. Click on the down arrow next to the semaphore.
   b. In the **Priority list** area, define the order using the arrows.
4. One robot controller is defined as the semaphore master. The semaphore master can be changed:
   a. Click on the down arrow next to the semaphore.
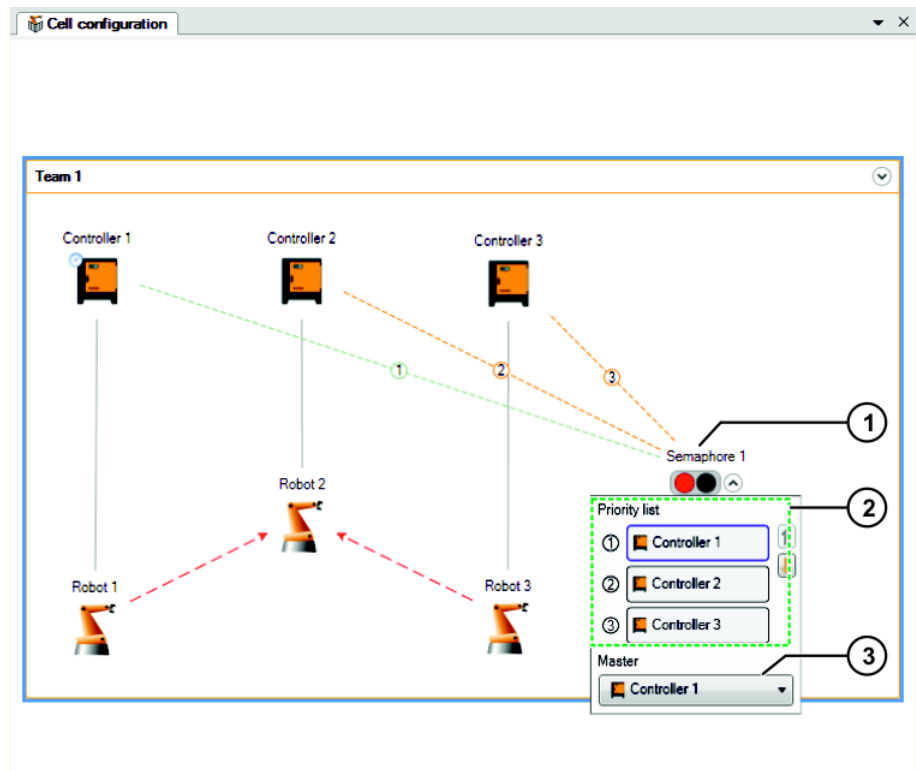   b. Select the desired robot controller in the **Master** area.

Fig. 11-7: Configuring semaphores

Item	Description
1	Semaphore display
2	**Priority list** This order can be changed using the arrows.
3	Semaphore master

12 Programming

12.1 Creating a program

Precondition
- With use of a KR C4 controller: the **KRL Templates** catalog has been inserted in the **Catalogs** window.
- With use of a VKR C4 controller: the **VW Templates** catalog has been inserted in the **Catalogs** window.

Procedure
1. Expand the tree structure of the robot controller on the **Files** tab in the **Project structure** window.
2. Select the required template in the **KRL Templates** or **VW Templates** catalog and drag it onto a node in the tree structure. The program file is inserted in the tree structure.

   The file can now be edited with the KRL Editor.

12.2 Importing a program

Description   Files in the formats SRC, DAT, SUB and KRL can be imported.

Procedure
1. Expand the tree structure of the robot controller on the **Files** tab in the **Project structure** window.
2. Right-click on the node in which the program is to be created and select **Add external file** from the context menu.
3. Navigate to the directory where the file is located that is to be imported.
4. Select the file and confirm with **Open**. The file is inserted in the tree structure.

   The file can now be edited with the KRL Editor.

12.3 Displaying the variable declarations in a file

All the KRL variables that are declared in a particular file can be displayed in a list overview. With SRC files, the variables from the corresponding DAT file are always displayed at the same time, and vice versa.
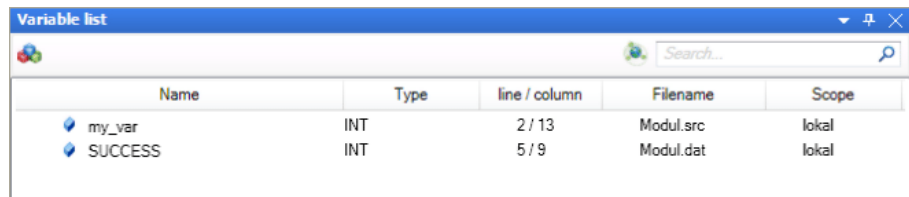
Procedure
1. Only if the **Variable list** window is not yet displayed: Display it by selecting the menu sequence **Window** > **Variable list**.
2. Open the file in the KRL Editor, or, if it is already open, click on the tab for the file.
3. The variable list now displays all the variables that are declared in this module (SRC file and corresponding DAT file).
4. If necessary, a variable can be selected in the KRL Editor as follows:
   - Double-click on the line in the search results.
   - Or: Right-click on the line and select **Go to…** from the context menu.
   - Or: Select the line and press the Enter key.

A search function is available in the **Variable list** window, which allows a search to be carried out for local variables within the current file:

- Enter the variable name or part of the name in the search box. The search results are displayed immediately.

If the cursor is positioned in the search box, this can be emptied by pressing ESC.

Description

Fig. 12-1: "Variable list" window

Clicking on a column sorts the list by this column.

Button	Name / description
	Groups the variables by local subfunctions Button is pressed: The display is sorted by file type. (Within this sort mode, it is also possible to sort by column.) Button is not pressed: The display is not sorted by file type.

12.4 Searching and replacing within files

Description

WorkVisual provides a search function which allows the text of all files in the entire project to be searched. It is also possible to search an individual file or a selected area within a file. The range of the search can be selected in the search window.

Instead of searching, it is also possible to search and replace.

The search (or search and replace) function can be called from any place in the project, whatever workspace or editor is currently in focus.

Procedure

1. If an individual file is to be searched, this should be opened.
2. If an area within the file is to be searched, this should be selected.
3. Open the Search window: **Ctrl+F**

   Or: Open the window for Search and Replace: **Ctrl+H**
4. Make the desired settings and click on **Search**, or on **Replace** or **Replace all**.

12.5 KRL Editor

12.5.1 Opening a file in the KRL Editor

Precondition

- This is a file format that can be edited with the KRL Editor.

  (>>> "File formats" Page 115)

Procedure

1. Double-click on a file on the **Files** tab in the **Project structure** window.

   Or: Select the file and click on the **KRL Editor** button.

   Or: Select the file and select the menu sequence **Editors** > **KRL Editor**.
2. To close the file: Click on the "X" at top right.

The KRL Editor allows more than one file to be open simultaneously. If required, they can be displayed side by side or one above the other. This provides a convenient way of comparing contents, for example.

To display the files side by side:

1. Right-click in the title bar of the file in the KRL Editor. Select **New Vertical Tab Group** from the context menu.

2. To display the files one behind the other again: Right-click in the title bar of a file in the KRL Editor. Select **Move Next** or **Move Previous** from the context menu.

To display the files one above the other:

1. Right-click in the title bar of the file in the KRL Editor. Select **New Horizontal Tab Group** from the context menu.

2. To display the files one behind the other again: Right-click in the title bar of a file in the KRL Editor. Select **Move Next** or **Move Previous** from the context menu.

File formats   The KRL Editor is used primarily to edit files containing KRL code:

- SRC
- DAT
- SUB

In addition, the KRL Editor can be used to edit the following file formats:

- ADD
- BAT
- CONFIG
- CMD
- DEL
- INI
- KFD
- KXR
- LOG
- REG
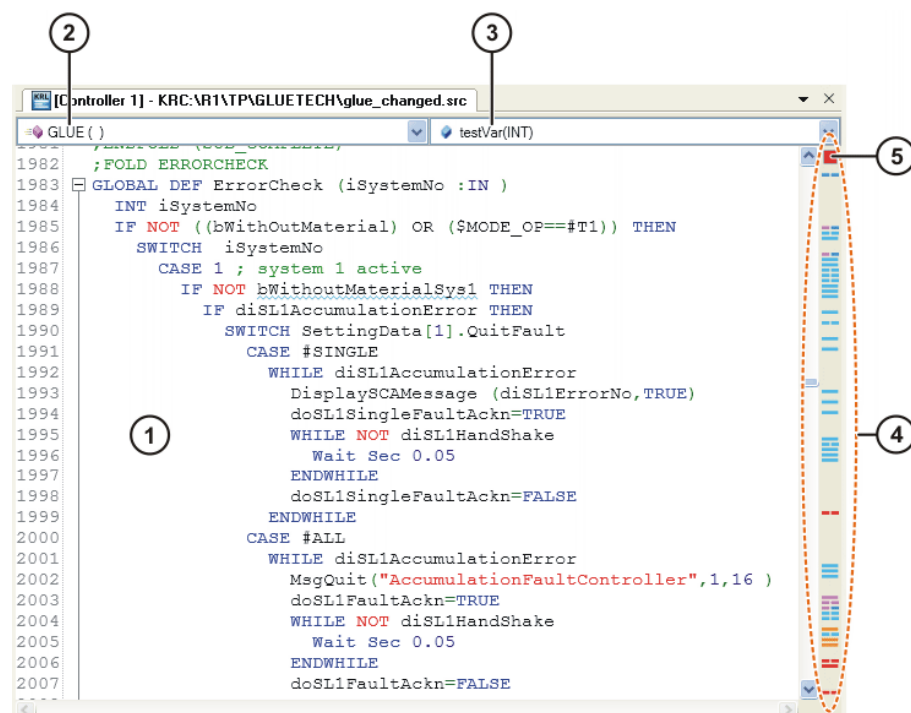- TXT
- XML

12.5.2   KRL Editor user interface

Fig. 12-2: KRL Editor user interface

Item	Description
1	Program area
	The code is entered and edited here. The KRL Editor provides numerous functions to support the programmer with this.
2	List of the subprograms in this file
	To go to a subprogram, select it from the list: the cursor then jumps to the DEF line of this subprogram.
	If the file does not contain any subprograms, the list is empty.
3	List of variable declarations
	This list always refers to the subprogram that is currently selected in the list of subprograms. To go to a declaration, select the variable from the list: the cursor then jumps to the line containing the declaration of these variables.
	If there are no variable declarations, the list is empty.
4	Analysis bar
	The marks indicate errors or discrepancies in the code.
	■ Hovering with the mouse over a mark displays a tool tip with a description of the error.
	■ Clicking on the mark makes the cursor jump to the relevant place in the program. An automatic correction is offered for some errors/discrepancies.
	(>>> 12.5.9 "Quickfix correction" Page 120)
5	The square has the color of the most serious error currently present.
	If there are no errors/discrepancies, the square is green.

12.5.3 Zooming in/out

Procedure
1. Click anywhere in the KRL Editor.
2. Hold down the CTRL key and move the mouse wheel.
   - ■ Mouse wheel up: zoom in
   - ■ Mouse wheel down: zoom out

12.5.4 Configuring KRL Editor

Preparation
Only necessary if it is desirable to see a preview of the effect of the settings:
1. Open a file in the KRL Editor.
2. To see a preview of the colors for the marks: select any point in the file.
   (Nothing can be selected in the file while the **Options** window is open.)

Procedure
1. Select the menu sequence **Extras** > **Options**. The **Options** window is opened.
2. On the left of the window, open the **Text editor** folder. In the folder, select a subitem.
   The corresponding settings are now displayed on the right of the window.

	If the mouse pointer is moved over a field, a description is displayed for this field at the bottom of the window.

3. Make the desired changes.

The changes can be seen immediately if a file is currently open in the KRL Editor (e.g. if spaces are revealed or hidden).

4. Confirm with **OK**. The changes are saved.

Or discard the changes with **Cancel**.

The color settings can be reset to the default settings at any time. The **Reset** button for this is situated on the corresponding page in the **Options** window. (At the bottom of the page; scrolling is required.)

12.5.5 Edit functions

12.5.5.1 General edit functions

Select

■ To select an area: click where the selection should begin and hold down the left-hand mouse button. Drag the mouse until the desired area is selected, and then release the mouse button.

> If the ALT key is additionally held down during selection, it is possible to select a rectangular area.

■ To select a line: click on the line number.

Edit

Standard edit functions can be called via the context menu. These include:

■ **Cut**, **Paste**, **Copy**, **Delete**

■ **Undo**, **Redo**

■ **Search …** (>>> 12.4 "Searching and replacing within files" Page 114)

In addition, the following commands are available in the context menu.

> ■Not all commands are available for every file format.
>
> ■Commands which refer to a selected area affect the whole file if no area is selected.

Menu item	Description
Edit > **Make upper case**	Converts all the lower-case letters in the selected string into upper-case letters.
Edit > **Make lower case**	Converts all the upper-case letters in the selected string into lower-case letters.
Edit > **Title case**	Converts all the initial letters in the selected string into upper-case letters.
Edit > **Convert tabs into spaces**	Replaces the tabs in the selected string with spaces. **Note:** The number of spaces corresponding to one tab can be configured in the parameter **Indent size**.
Edit > **Convert spaces into tabs**	Replaces the spaces in the selected string with tabs.
Edit > **Indent**	Inserts additional spaces at the beginning of every line in the selected area. **Note:** The number of spaces inserted can be configured in the parameter **Indent size**.
Edit > **Remove leading whitespace**	Removes all the leading spaces from the lines in the selected area.
Folds > **Expand all**	Opens all folds in the currently displayed file.
Folds > **Collapse all**	Closes all folds in the currently displayed file.

Menu item	Description
Format	Indentations, line breaks, etc., are adapted throughout the file to conform to the standard. The applicable standard depends on the file format.
Comment selection	Uncomments the line.
Uncomment selection	Comments the line out.
Rename	(>>> 12.5.5.2 "Renaming a variable" Page 118)
Go to declaration	(>>> 12.5.7 "Jumping to the declaration of a variable" Page 120)
Insert snippet	(>>> 12.5.5.4 "Snippets – Fast entry of KRL instructions" Page 119)

12.5.5.2 Renaming a variable

Description
A variable name can be changed in a single action at all points where it occurs.

This is also possible if the variable is declared in a DAT file and used in various SRC files.

Precondition
- The files in which the variable name occurs do not contain any syntax errors.

  Automatic renaming cannot be executed in a file containing syntax errors.

Procedure
1. Select the desired variable at any point.
2. Right-click and select **Rename** from the context menu.
3. A window opens. Change the name and confirm with **OK**.

12.5.5.3 Auto-complete

An auto-complete function is available in the KRL Editor.

When entering code, a list containing the following elements is displayed automatically:

- KRL keywords
- Known variable names
- Known function names
- Known user-specific data types (STRUC or ENUM)
- Snippets (>>> 12.5.5.4 "Snippets – Fast entry of KRL instructions" Page 119)

Those elements that are compatible with the characters already entered are shown at the top of the list. These elements are also prioritized according to their frequency of use, i.e. the selection is dynamically adapted to the user's actions.

If required, an element in the list can be selected and inserted in the program text using the Enter key. This makes it unnecessary to type complex variable names, for example.

> Navigation in the "Auto-complete" list:
> - Scroll
> - Or: Type the first letter of the desired element. The marker jumps to the relevant position.

12.5.5.4 Snippets – Fast entry of KRL instructions

Description   A fast entry function is available in the KRL Editor for common KRL instructions.

To program a FOR loop, for example, it is not necessary to enter the entire syntax `FOR … = … TO … STEP …`. Instead, the instruction can be selected from the "Auto-complete" list. All that is then required is to fill out the variable positions in the syntax manually.

Procedure   On starting to type the code, the "Auto-complete" list is shown. The required instruction is generally already selected.

1.   Accept the selected instruction from the "Auto-complete" list using the Enter key. Or double-click on a different instruction.
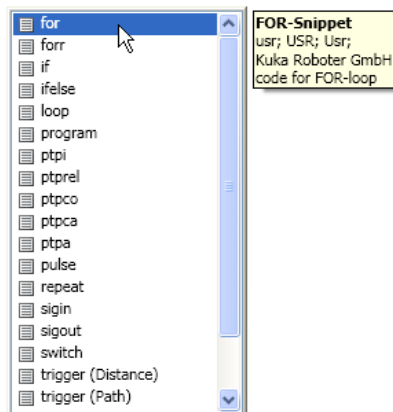
Fig. 12-3: Accept with the Enter key, or double-click

2.   The KRL syntax is inserted automatically. The first variable position is highlighted in blue. Enter the desired value.
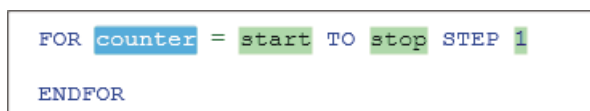
Fig. 12-4: The first variable position is highlighted in blue

3.   Press the TAB key to jump to the next variable position. Enter the desired value.
4.   Repeat step 3 for all other variable positions.
5.   To finish editing, press the Enter key.

The snippet list can also be called separately: Right-click and select **Insert snippet** from the context menu.

It is also possible to enter a snippet as follows: Type in the abbreviation and press the TAB key.

(The abbreviation can be determined by calling the snippet list. Select the instruction. A tool tip is displayed. The 2nd line contains the possible abbreviations.)

12.5.6   Folds

Description   The content of the KRL Editor can be structured with folds just like any normal KRL program.

Fig. 12-5: Closed fold

Fig. 12-6: Opened fold

Procedure   To open a fold:

- Double-click on the box of the closed fold.
- Or: Click on the plus sign.

To close a fold:

- Click on the minus sign.

To open or close all folds:

- Context menu: **Folds** > **Expand all** or **Collapse all**

12.5.7 Jumping to the declaration of a variable

Procedure   1. Place the cursor in the variable name, or directly in front of the first letter or directly after the last letter.
2. Right-click and select **Go to declaration** from the context menu.

12.5.8 Displaying all uses of a variable

Procedure   1. Only if the **Find uses** window is not yet displayed:

Select the menu sequence Window > **Find uses**.
2. Place the cursor in the variable name, or directly in front of the first letter or directly after the last letter.
3. Right-click and select **Find uses** from the context menu.

A tab with the name **Uses of [***Variable name***]** appears in the **Find uses** window. All uses are listed there in detail (file with path, line number, etc.)
4. If required: Double-click on a line in the list. The corresponding position in the program is highlighted.

Rename, for example, is now possible.

12.5.9 Quickfix correction

Wavy red lines in the code and marks in the analysis bar indicate errors or discrepancies in the code.

For some of these errors/discrepancies, an automatic correction function – the "Quickfix" – is offered. A Quickfix light bulb is displayed. Via the arrow button

next to the light bulb, the user can display various possible solutions and select one.

Fig. 12-7: Quickfix light bulb

12.5.9.1 Correcting or automatically declaring undeclared variables

Description   Undeclared variables are displayed as follows:

- With a wavy red line in the code
- With a red mark in the analysis bar

The red color can also refer to a different error, however. If it refers to an undeclared variable, the following tool tip is displayed if you hover with the mouse over the wavy line or mark: *The declaration of variable [name] was not found*.

Procedure   1. Place the cursor in the name underlined with the wavy line, or directly in front of the first letter or directly after the last letter.

Or: Click on the mark in the analysis bar.

The Quickfix light bulb is now displayed next to the variable name.

2. Check if the variable name has been written incorrectly (differently from in the declaration).

- If so: correct. The wavy red line / mark disappears. No further steps are necessary.
- If not: continue with the next step.

3. Move the mouse pointer onto the Quickfix light bulb. An arrow is displayed next to the light bulb.

Click on the arrow. The following options are displayed:

- *Declare the variable locally*
- *Declare the variable in the data list*

4. Click on the desired option.

5. Only with *Declare the variable in the data list*: The data list is opened.

Open the fold BASISTECH EXT.

6. A snippet for the variable declaration has been automatically inserted. The expected data type is highlighted in blue. The declaration is followed by the comment: *; This variable is for ....*

- Accept or change the data type, as required.
- Press the TAB key to jump to the comment. Edit the comment if required.

> This comment is displayed in the tool tip of the "Auto-complete" list if the variable is selected there.

12.5.9.2 Removing unused variables

Description   Unused variables are displayed as follows:

- With a wavy blue line in the code
- With a blue mark in the analysis bar

Hovering with the mouse over the wavy line or mark displays a tool tip with a description.

Procedure	1. Place the cursor in the name underlined with the wavy line, or directly in front of the first letter or directly after the last letter.

Or: Click on the mark in the analysis bar.

The Quickfix light bulb is now displayed next to the variable name.

2. Move the mouse pointer onto the Quickfix light bulb. An arrow is displayed next to the light bulb.

Click on the arrow. The following options are displayed:

■ *Remove declaration*

■ *Comment out declaration*

3. Click on the desired option.

12.5.9.3 Standardizing the upper/lower case in a variable name

Description If the use of upper/lower case in a variable name is not uniform in the declaration and in its other occurrences, this is displayed as follows:

■ With a light blue wavy line in the code

■ With a light blue mark in the analysis bar

Hovering with the mouse over the wavy line or mark displays a tool tip with a description.

Procedure 1. Place the cursor in the name underlined with the wavy line, or directly in front of the first letter or directly after the last letter.

Or: Click on the mark in the analysis bar.

The Quickfix light bulb is now displayed next to the variable name.

2. Move the mouse pointer onto the Quickfix light bulb. An arrow is displayed next to the light bulb.

Click on the arrow. The following options are displayed:

■ *Change this use to [name as in declaration]*

■ *Change declaration to [name as at this point in the program]*

3. Click on the desired option.

12.5.10 Creating user-specific snippets

Description Users can create their own snippets. For this, the required properties must be saved in a file with the SNIPPET format. This file must then be imported in WorkVisual. The snippet is then available in the KRL Editor.

A template for a SNIPPET file is provided on the WorkVisual CD in the DOC directory.

> **i** A snippet that has been imported into WorkVisual is also available in OptionPackageEditor and vice versa. A precondition for this is that WorkVisual and OptionPackageEditor are running on the same PC and the same user is logged on.

Procedure Once the SNIPPET file has been created, it must be imported as follows:

1. Select the menu sequence **Extras** > **Import snippet from file …**. A window opens.

2. Navigate to the directory where the SNIPPET file is located and select it. Click on **Open**.

The snippet is now available in the KRL Editor.

Example 1 A snippet is to be created to insert the following code structure:

```
MYTHING true

ENDTHING
```

Fig. 12-8: Code to be inserted by the snippet

The snippet is to have the name "User" in the snippet list, and the tool tip should contain the information shown here:

Fig. 12-9: Desired snippet
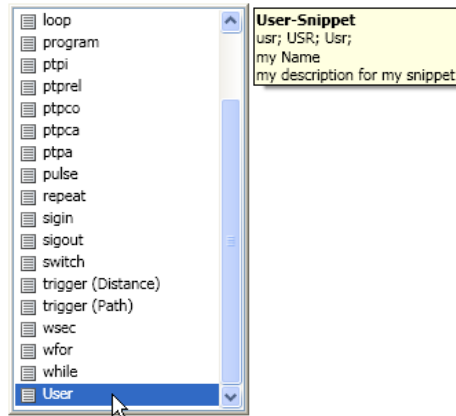
The SNIPPET file must have the following format:

```
2   <CodeSnippets>
3       <CodeSnippet Format="1.0.0">
4           <Header>
5
6               <!--Is displayed as header in the static ToolTip(on the right side) of the completion window-->
7               <Title>User-Snippet</Title>
8
9               <!--Is displayed in the completion window-->
10              <Text>User</Text>
11
12              <!--These shortcuts can be used-->
13              <Shortcut>usr</Shortcut>
14              <Shortcut>USR</Shortcut>
15              <Shortcut>Usr</Shortcut>
16
17              <!--For these file extensions the snippet will be shown -->
18              <Extensions>.src .sub</Extensions>
19
20              <!--Is displayed as description in the static ToolTip(on the right side) of the completion window-->
21              <Description>my description for my snippet</Description>
22
23              <!--Is displayed as author in the static ToolTip(on the right side) of the completion window-->
24              <Author>my Name</Author>
25
26              <!--Specifies the type of the snippet-->
27              <SnippetTypes>
28                  <SnippetType>Expansion</SnippetType>
29                  <SnippetType>SurroundsWith</SnippetType>
30              </SnippetTypes>
31          </Header>
32
33          <FileExtensions/>
34          <Snippet>
35              <Declarations>
36                  <Literal>
37                      <ID>element</ID>
38                      <ToolTip>my tooltip for this element</ToolTip>
39                      <Default>true</Default>
40                  </Literal>
41              </Declarations>
42              <Code Language="KRL">
43                  <![CDATA[MYTHING $element$
44  $end$$selection$
45  ENDTHING]]>
46              </Code>
47          </Snippet>
48      </CodeSnippet>
49  </CodeSnippets>
```

Fig. 12-10: Structure of the SNIPPET file

Line	Description
3 … 48	Section for 1 snippet A SNIPPET file may contain a number of these sections, i.e. a number of different snippets.
7	Title displayed in the tool tip
10	Name displayed in the snippet list **Note:** The "Auto-complete" function reacts to this character string, i.e. if this string is entered in the program, the "Auto-complete" list is shown and the corresponding snippet is selected.
13 … 15	Shortcuts for this snippet
18	The snippet is only displayed in the list in files with this extension
21	Description displayed in the tool tip
24	Name of the author displayed in the tool tip

Line	Description
27 … 30	This defines the ways in which the snippet can be inserted. ■ `Expansion`: The snippet is inserted at the current cursor position. ■ `SurroundsWith`: Before the snippet is inserted, program lines can be selected in the KRL Editor. The snippet is then automatically inserted so that it surrounds these lines. The exact position of these lines within the snippet is defined by the placeholder `$selection$`.
37	Placeholder occurring in lines 43 … 45, to which lines 38 and 39 refer
38	Tool tip displayed for this placeholder
39	Default value for the placeholder
43 … 45	Program text inserted by the snippet The text consists of hard text and/or placeholders. ■ `$selection$`: See description for `SurroundsWith`. ■ `$end$`: This placeholder defines the position of the cursor after insertion of the snippet has been completed with the Enter key.

Example 2 An example of the <Snippet> section only:

Fig. 12-11: Code inserted by the snippet

Fig. 12-12: Structure of the SNIPPET file

12.6 Working online with the KRC Explorer

This functionality is only available for robot controllers with KUKA System Software version 8.3.15 or higher or with VW System Software version 8.3.11 or higher.

With the KRC Explorer, changes can be made directly on the file system of the robot controller without a project having to be transferred for this purpose.

For the editing procedure, WorkVisual creates a local copy of the file system; this is where the changes are carried out. The changes can be synchronized with the robot controller at any time; this includes individual changes.

The file system of the robot controller is referred to below as the working directory.

12.6.1 Opening the working directory in the KRC Explorer

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- **Programming and diagnosis** workspace

Procedure
1. Click on the **Create connection** button.

   Alternatively: Right-click in the empty space in the KRC Explorer and select **Create connection** from the context menu.

   A window for selecting the robot controller is opened. In this window, it is possible to filter by type and version of the controller. There is also a search function available, which allows a search for the name of the controller to be carried out.
2. Select the desired robot controller and confirm with **OK**. The working directory of the robot controller is loaded and displayed in the KRC Explorer.

12.6.2 KRC Explorer user interface
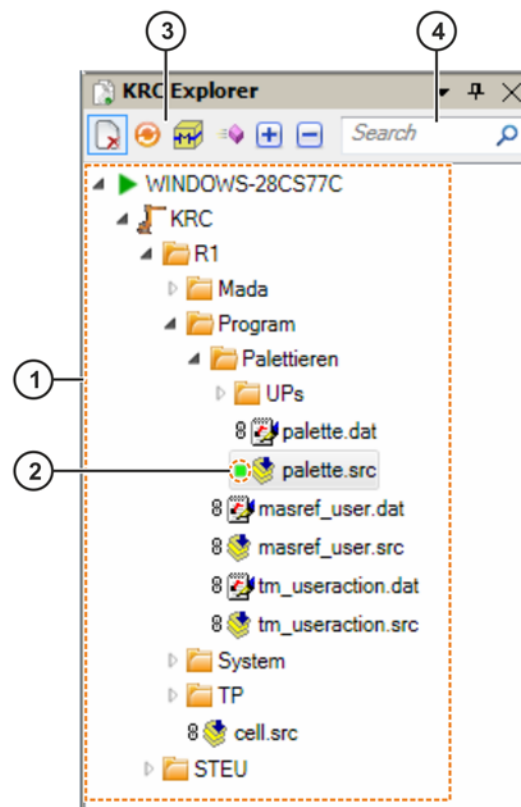
Fig. 12-13: KRC Explorer user interface

Item	Description
1	Working directory of the robot controller
2	The icon indicates the current status of the folder or file. Hovering with the mouse over the icon displays a tool tip with a description of the status. Additional information is displayed for certain statuses. If a file has been moved, for example, the original path is displayed in the tool tip. (>>> 12.6.2.2 "Icons in KRC Explorer" Page 127)
3	Button bar (>>> 12.6.2.1 "Button bar in KRC Explorer" Page 127)
4	The search box can be used to search for file and folder names.

12.6.2.1 Button bar in KRC Explorer

Button	Description
	Shows or hides deleted files and folders.
	Monitors changes on the file system of the robot controller. This icon is displayed after the name of modified files.
	Module view Shows SRC and DAT files combined as a module, like on the robot controller.
	Shows all subprograms of a module in the working directory. Double-clicking on a subprogram shows the point in the editor at which the subprogram was defined.
	Expands the selected node and all subnodes.
	Collapses the selected node and all subnodes.

12.6.2.2 Icons in KRC Explorer

Icon	Description
	The file has been changed.
	Module containing an SRC file and/or a DAT file.
	Subprogram in an SRC file
	The file or folder is unchanged.
	The file or folder has been renamed.
	The file has been renamed and changed.
	The file or folder has been moved. This icon is also displayed if the file was renamed before or after being moved.
	The file or folder has been deleted. A file that has been deleted can no longer be changed.
	The file or folder has been added to the working directory. This icon is also displayed if the file was changed, renamed or moved after being added.

Icon	Description
	The file has been changed.
	The file has been moved and changed.

Icons in the module view:

Icon	Description
	The module has been changed and moved.
	The module has been changed and renamed.
	The SRC or DAT file has been changed and the other one has been deleted.
	The SRC or DAT file has been changed and the other one remains unchanged.
	The SRC or DAT file has been deleted and the other one remains unchanged.
	The SRC or DAT file has been moved and the other one remains unchanged.
	The SRC or DAT file has been renamed and the other one remains unchanged.
	The SRC or DAT file has been changed and the other one has been renamed and changed.
	The SRC or DAT file has been changed and renamed and the other one remains unchanged.

Icons in the debugging mode:

Icon	Description
	The program is linked.
	The program is not selected.
	The program has been selected and terminated.
	The program has been selected and is being executed.
	The program has been selected and stopped.
	The program has been selected and reset.

12.6.3 Configuring KRC Explorer

Procedure
1. Select the menu sequence **Extras** > **Options...**. The **Options** window is opened.
2. On the left of the window, in the **Online workspace** folder, select the subitem **KRC Explorer**.
   The corresponding settings are now displayed on the right of the window.
3. Select the directory in which working directories are to be saved by default.
4. Click on **OK** to confirm. The settings are saved.

12.6.4 General edit functions

Standard edit functions can be called via the context menu. These include:

- **Cut**, **Copy**, **Paste**, **Rename**, **Delete**

In addition, the following commands are available in the context menu.

Menu item	Description
Editor	Opens the file in the KRL Editor.
Open with...	Opens the file in the selected program.
Start debugging	Starts the debugging mode.
Stop debugging	Ends the debugging mode.
Open in Windows Explorer	Opens the selected directory or the directory of the selected file in Windows Explorer.
Add...	Creates a file in the format of the selected template under the selected folder. (>>> 12.6.5 "Creating a new file with template" Page 129)
Save as template	Saves the selected file as a template.
Disconnect VRC	Terminates the connection to the robot controller. The working directory is deleted.
New folder...	Creates a new folder under the selected folder.
Compare with controller version	Compares the selected file with the file on the robot controller.
Compare with...	(>>> 12.6.9 "Comparing files" Page 131)

12.6.5 Creating a new file with template

Procedure
1. In the working directory, right-click on the folder in which the file is to be created.
2. Select **Add...** from the context menu. The **Select template** window opens.
3. Select the desired template.

    The KUKA templates, templates from installed option packages and the templates of the current robot controller are available for choice as the template.
4. Enter a name in the **Name** box and confirm with **OK**.

12.6.6 Transferring changes to the robot controller

Description
This procedure is used to transfer all changes or only selected changes in the working directory from WorkVisual to the real robot controller.

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- "Expert" user group
- T1 or T2 mode
- **Programming and diagnosis** workspace

Procedure
1. To transfer all changes: Select the root node of the working directory.

    To transfer selected changes: Select the file or folder that is to be transferred.
2. Click on the **Transfer changes to the controller.** button.

    Alternatively: Right-click and select **Transfer changes** from the context menu.

    The changes are checked for conflicts. A message is displayed if conflicts are detected. Conflicts must be resolved; otherwise, the changes cannot

be transferred. If there are no conflicts, the changes are shown in an overview.

> **i** If a file contains different changes in the working directory in WorkVisual and on the real robot controller, there are 3 possibilities:
> - Make no changes
>
> ■ Overwrite the file on the real robot controller with the file in the working directory
>
> ■ Merge the files
>
> An external editor is required to merge the files.
> (>>> 6.27.4 "Configuring additional editors" Page 61)

3. If you do not wish to transfer all the displayed changes: Deactivate the check boxes of the relevant changes.

4. Confirm with **OK**. The KUKA smartHMI displays the request for confirmation *The user […] has requested remote access to this controller. Do you want to allow remote access?*.

5. Confirm the request with **Yes**. The changes will be transferred. The message *The user […] has remote access to this controller.* is displayed on the smartHMI. This access remains active until the message is acknowledged with **Revoke**.

> **NOTICE** After changes have been transferred, all programs must be tested in T1 mode. Damage to property may otherwise result.
> This applies particularly if the transfer procedure was interrupted and not all changes were carried out.

12.6.7 Loading changes from the robot controller

Description   This procedure is used to load all changes or only selected changes from the robot controller. Changes in the working directory that have not yet been transferred to the robot controller are displayed and can be undone.

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- **Programming and diagnosis** workspace

Procedure
1. Either: Right-click on the root node of the working directory and select **Load files from controller**. All the changes are shown in an overview.

   Or: Right-click on a folder in the working directory and select **Load files from controller**. The selected folder, the files contained in it and the changes to them are shown in an overview.

2. Activate or deactivate the check boxes of the changes as desired and confirm with **OK**. The selected changes will be carried out.

12.6.8 Restoring the state of the robot controller

Description   This procedure is used to discard all local changes in the working directory and to restore the state of the robot controller.

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- **Programming and diagnosis** workspace

Procedure
1. Right-click on the root node of the working directory and select **Establish controller state**.

2. Answer the request for confirmation with **Yes**. The state of the real robot controller is restored and all changes are overwritten.

12.6.9 Comparing files

Description

This procedure is used to compare a file in the working directory in WorkVisual with the same file on the robot controller. The file can also be compared with a file on the local hard drive.

> Any appropriate software can be used for the comparison. It must be installed and configured in WorkVisual, however.
> (>>> 6.27.4 "Configuring additional editors" Page 61)

Precondition

■ **Programming and diagnosis** workspace

Only for comparison with the robot controller:

■ Network connection to the real robot controller
■ The real robot controller and the KUKA smartHMI are running.

Procedure

Comparison with the file on the robot controller:

■ Either: Right-click on the file in the working directory and select **Compare with controller version** from the context menu. The files are opened with the editor that has been configured as the default editor for comparisons.
■ Or: Select **Compare with...** > **File on controller** from the context menu and select the desired editor. The files are opened with the desired editor.

Comparison with a file on the local hard drive:

1. Right-click on the file in the working directory, select **Compare with...** > **Local file** from the context menu and select the desired editor. A window opens.
2. Navigate to the path where the file for comparison is located.
3. Select the file and confirm with **Open**. The files are opened with the desired editor.

12.7 Debugging programs

WorkVisual provides a debugging mode with which errors can be found in running programs. Programs with subprograms can also be debugged.

12.7.1 Overview of the debugging mode
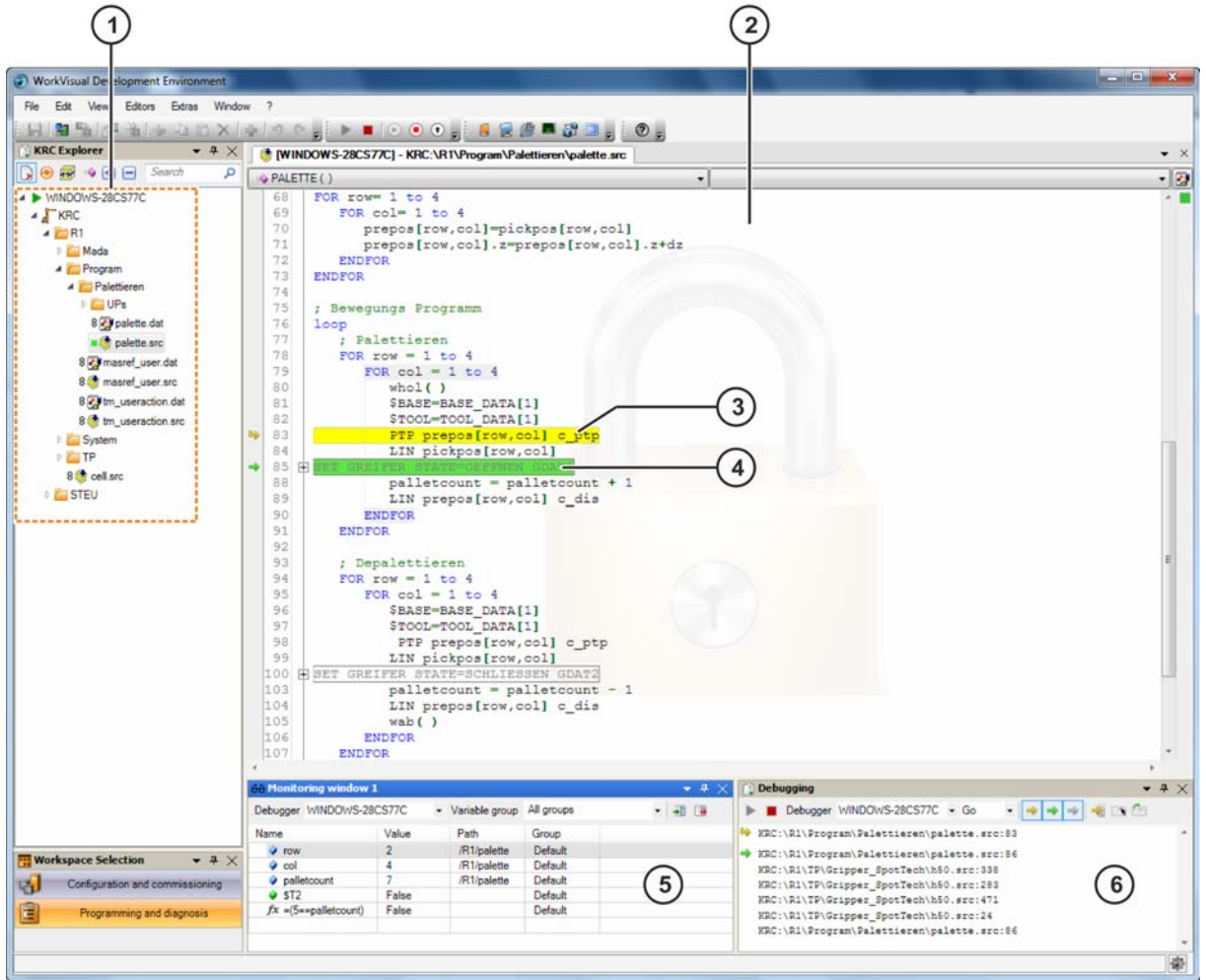
Fig. 12-14: Debugging mode

Item	Description
1	Working directory in the debugging mode
2	Program running in the debugging mode
3	Current position of the block pointer
4	Current position of the advance run pointer
5	Monitoring window (>>> 12.7.3 "Monitoring variables" Page 134)
6	Debugging window (>>> 12.7.2 "Starting the debugging mode" Page 133)

Point-er	Description
⇨	Block pointer The end point is being approached.
→	Advance run pointer
⇨	The block pointer is located in a subprogram.
⇨	The block pointer and advance run pointer are located in the same line. The end point has been reached with exact position-ing.

Point-er	Description
⇨	The block pointer and advance run pointer are located in the same line. The end point is being approached.
⇨	Block pointer
	The end point has been reached with exact positioning.
⬆	The block pointer is located in a program line higher up in the program.
⬇	The block pointer is located in a program line lower down in the program.
⬆	The block pointer is located in a subprogram. The current program line is located higher up in the program.
⬇	The block pointer is located in a subprogram. The current program line is located lower down in the program.

Button	Description
▶	Starts the debugging mode.
■	Ends the debugging mode.

The following buttons are only available when an OPS is used:

Button	Description
▶	Starts the program.
■	Stops the program.
↑	Resets the program.

12.7.2 Starting the debugging mode

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- "Expert" user group
- T1 or T2 mode
- **Programming and diagnosis** workspace
- The working directory is open in the KRC Explorer and the state corresponds to that of the real robot controller.

Procedure
1. Only if the **Debugging** window is not yet displayed: Display it by selecting the menu sequence **Window** > **Debugging**.
2. Right-click on the working directory in the KRC Explorer and select **Start debugging**.

   Alternatively: Click on the ▶ button.
3. Select and start the desired program on the smartHMI.
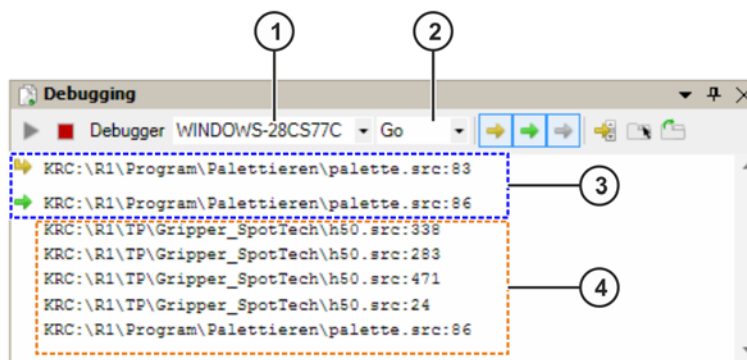
Fig. 12-15: "Debugging" window

Item	Description
1	The robot controllers whose working directory has been loaded into WorkVisual are displayed here.
2	The current program run mode is displayed here. A different program run mode can be selected. This setting is only relevant if an OPS is used.
3	These lines indicate the programs and lines in which the block pointer and advance run pointer are currently located.
4	List of calls These lines indicate which programs or functions within programs are called. Double-clicking on a line jumps directly to the relevant point in the program. In this example, a function is called in line 283 in the program h50.SRC. From this function, a function or a program in the program palette.SRC is called in line 338.

Button	Description
▶	Starts the debugging mode.
■	Ends the debugging mode.
⇨	Indicates the block pointer in the editor.
➡	Indicates the advance run pointer in the editor.
⇨	Indicates in the editor that the block pointer is located in a subprogram.
⇥	Scrolls automatically to the point in the program at which the block pointer is located.
◻	Indicates subprograms already opened in the foreground if the block pointer is located in them.
◻	Opens an editor for each subprogram called.

12.7.3   Monitoring variables

Description   During debugging, the state of variables in the program can be monitored. There are 2 monitoring windows available for this. Mathematical calculations (e.g. variable + 5) and expressions as well as nested structures and arrays are also evaluated in these windows. If a value changes, it is displayed in red for a few seconds in the monitoring window.

The following variables can be monitored:

- Variables that have been declared globally (in the file $CONFIG.DAT or in a DAT file containing the keyword PUBLIC)
- Variables that have been declared locally in the DAT file (globally for the module)
- Variables that have been declared locally in the SRC file while the advance run pointer is located in the same program as the declaration (runtime data)

Precondition   ■   The debugging mode has been started.

Procedure
1. Display the monitoring window by selecting the menu sequence **Window** > **Monitoring window 1** or **Monitoring window 2**.
2. Right-click on the desired variable in the program and select **Monitor**.

    Alternatively: Click on an empty row in the **Name** column and enter the name of the desired variable. During entry, variables are suggested that are contained in the current program or have been globally defined.
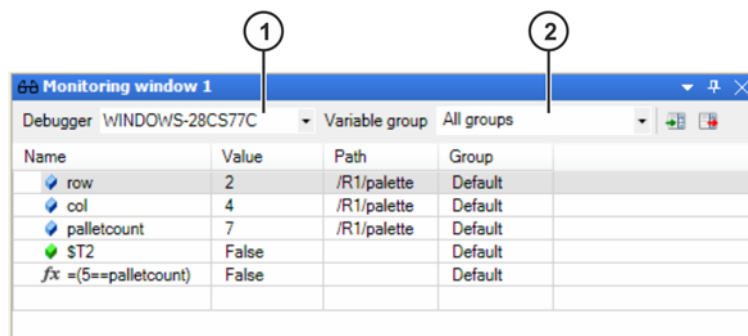
Fig. 12-16: Monitoring window

Item	Description
1	The robot controllers for which the debugging mode has been started are displayed here.
2	All the groups of variables present in the monitoring window are displayed here. A filter can be set for each group.

Button	Description
	Loads variables in the ConfigMon.INI format
	Saves variables in the ConfigMon.INI format

Icon	Description
	Variables that have been declared in either the DAT file or the SRC file
	Signals (e.g. inputs and outputs)
fx	Mathematical calculations and expressions

12.7.4   Ending the debugging mode

Procedure   ■   Right-click on the working directory in the KRC Explorer and select **Stop debugging**.

    Alternatively: Click on the ■ button.

13 Transferring and activating the project

13.1 Generating code

Description   When a project is transferred to the robot controller, the code is always generated first. This procedure can be used to generate the code separately and thus to check in advance whether generation runs without error.

The code is displayed on the **Files** tab of the **Project structure** window.

Automatically generated code is displayed in pale gray.
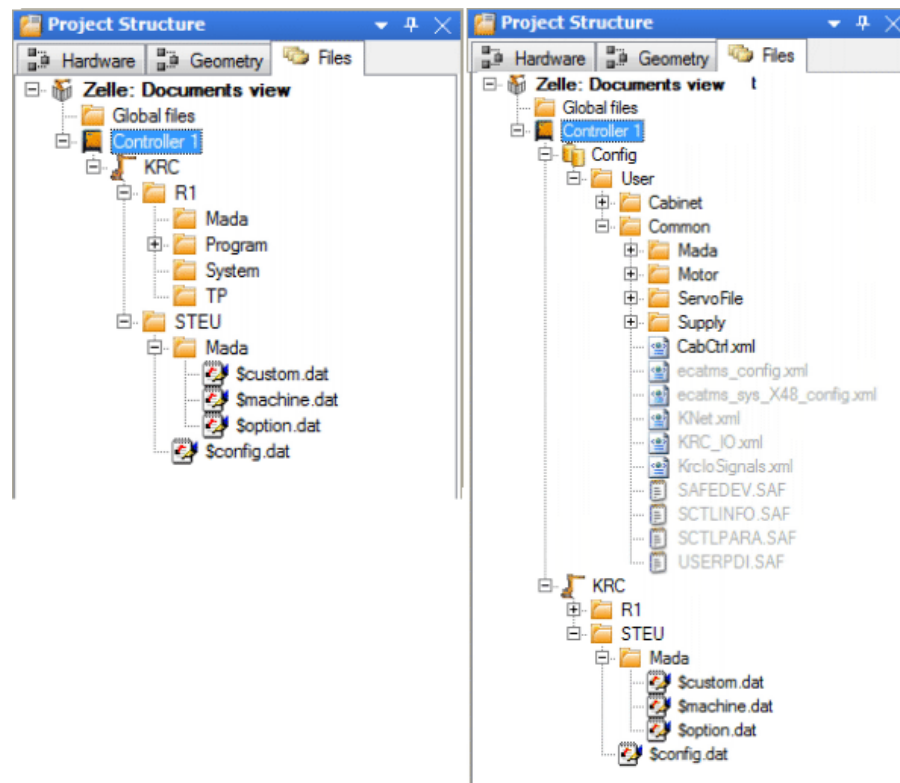
Fig. 13-1: Example of code generation: before – after

Procedure   ■ Select the menu sequence **Extras** > **Generate code**.

The code is generated. When the process is finished, the following messages are displayed in the message window: **The project <"{0}" V{1}> has been compiled. The results can be seen in the file tree.**

13.2 Pinning a project

Description   Projects that are present on the robot controller can be pinned. A project can be pinned directly on the robot controller or in WorkVisual.

Pinned projects cannot be changed, activated or deleted. They can be copied or unpinned, however. A project can thus be pinned e.g. to prevent it from being accidentally deleted.

Procedure   **Pinning in WorkVisual:**

1. Select the menu sequence: **File** > **Browse for project**. The **Project Explorer** is opened. On the left, the **Search** tab is selected.
2. In the **Available cells** area, expand the node of the desired cell. All the robot controllers of this cell are displayed.

3. Expand the node of the desired robot controller. All projects are displayed. Pinned projects are indicated by a pin symbol.

4. Select the desired project and click on the **Pin project** button. The project is pinned and labeled with a pin symbol in the project list.

> **i** Information about pinning on the robot controller can be found in the **Operating and Programming Instructions for System Integrators** for the KUKA System Software.

13.3 Assigning the robot controller to the real robot controller

Description

This procedure is used to assign every robot controller in the project to a real robot controller. The project can then be transferred from WorkVisual to the real robot controller.

Precondition

- A robot controller has been added in WorkVisual.
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.

If the project is subsequently to be transferred and also activated:

- The user group "Expert" or higher is selected on the real robot controller.
  Restriction: If the activation would cause changes in the area **Safety-relevant communication parameters**, the user group "Safety recovery" or higher must be selected.
- If the operating mode AUT or AUT EXT is selected on the real robot controller: The project contains only settings that affect KRL programs. If the project contains settings that would cause other changes, it cannot be activated.

Procedure

1. Click on the **Deploy…** button in the menu bar. The **WorkVisual Project Deployment** window is opened. The virtual robot controller from the project is displayed on the left-hand side. The target controller is displayed on the right-hand side. It is grayed out when a controller has not yet been selected.
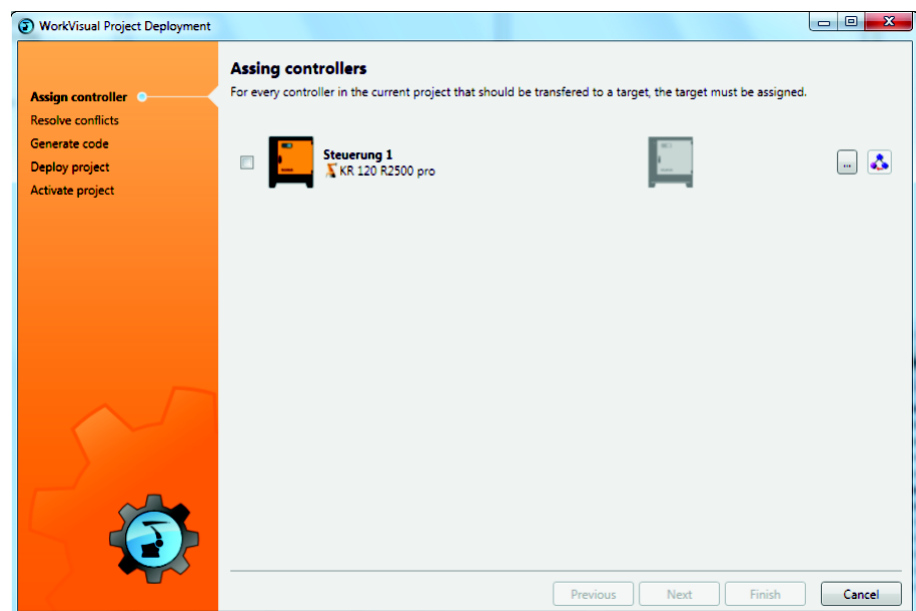
Fig. 13-2: Assigning the robot controller to the cell

2. On the left, activate the virtual cell by means of the check box. This cell must now be assigned to the real robot controller.

3. Click on the **…** button. A window opens. The filter is automatically set to only display controllers of the same type and version as the virtual controller. This setting can be changed.
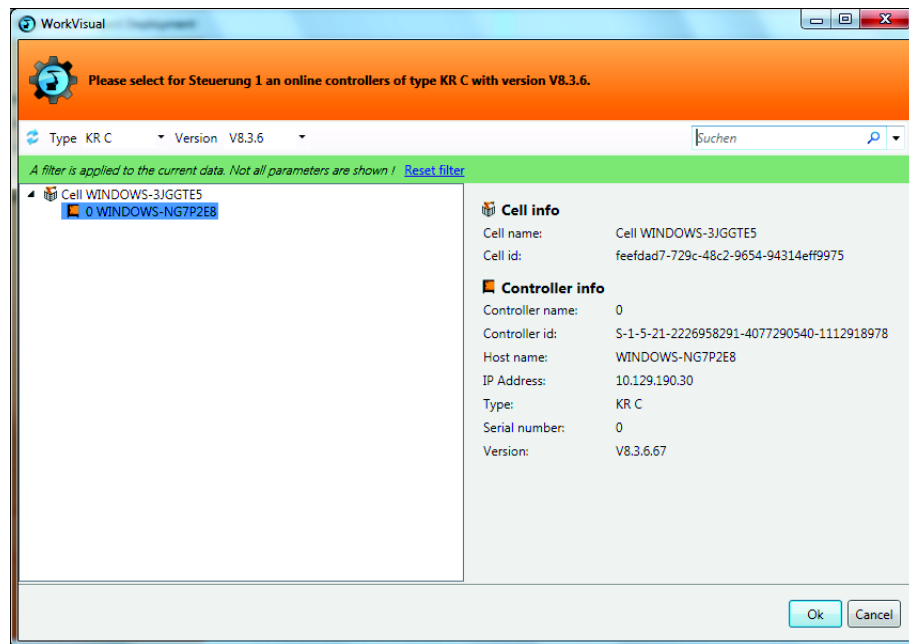
Fig. 13-3: Assigning a real robot controller to the virtual controller

4. Select the real robot controller and click on the **Ok** button. The real robot controller is assigned to the virtual robot controller. Once it is assigned, the real robot controller is displayed in color and with its name and IP address.
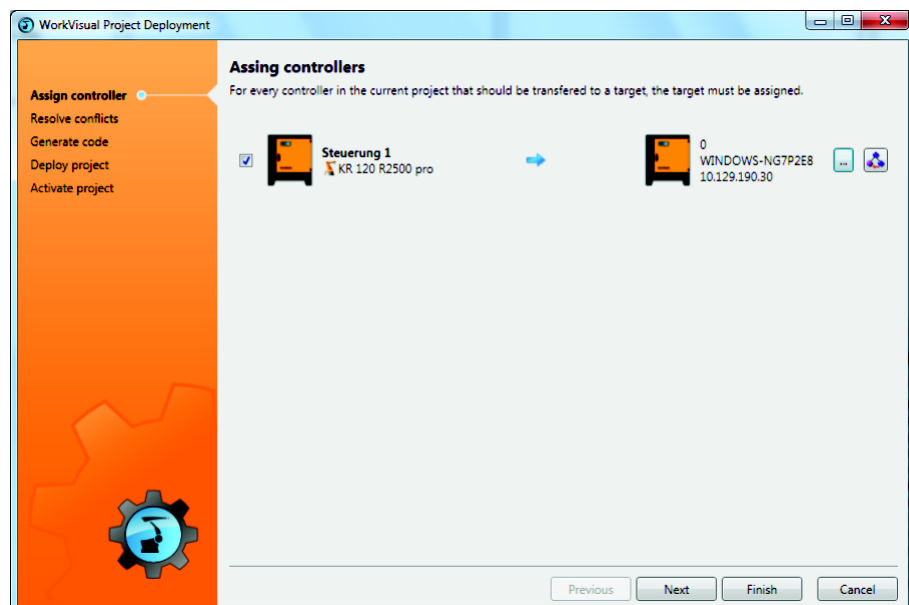
Fig. 13-4: Overview

5. If the project contains more than one robot controller, repeat steps 3 and 4 for the other robot controllers.

6. Click on **Next**. The assignment is checked for conflicts. A message is displayed if conflicts are detected. Conflicts must be resolved; otherwise, the project cannot be transferred. If no conflicts are present, the code is generated automatically.

7. The project can now be transferred to the robot controller.

Alternatively, the project can be transferred at a later point in time. To do this, click on **Cancel**: The assignment is saved and the **WorkVisual Project Deployment** window is closed.

13.4 Transferring the project to the robot controller

Description

This procedure is used to transfer the project from WorkVisual to the real robot controller.

> **i** If a project was transferred to the real robot controller at an earlier time and has not yet been activated then this will be overwritten if a further project is transferred.
>
> Transferring and activating a project overwrites a project of the same name that already exists on the real robot controller (after a request for confirmation).

Precondition

- The project has been assigned to the real robot controller.
- Assignment conflicts have been resolved.
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.

If the project is also to be activated:

- The user group Expert or higher is selected on the real robot controller.

  Restriction: If the activation would cause changes in the area **Safety-relevant communication parameters**, the user group "Safety recovery" or higher must be selected.

- If the operating mode AUT or AUT EXT is selected on the real robot controller: The project contains only settings that affect KRL programs. If the project contains settings that would cause other changes, it cannot be activated.

> **i** If one of the options KUKA.SafeOperation or KUKA.SafeRangeMonitoring is installed on the robot controller, different user groups may apply. Information can be found in the documentation for these options.

> **i** If a project is transferred containing an option package that has been inserted into the project without a window showing the changes being displayed, then the following procedure must be observed:
>
> 1. Transfer the project to the robot controller, but DO NOT activate it!
> 2. Install the option package on the robot controller.
>
>   Installation is performed in the normal manner. Information about this can be found in the documentation for the option package.
> 3. Transfer the active project from the robot controller to WorkVisual and carry out the user configuration.
> 4. Transfer the project back to the robot controller.
> 5. Activate the project on the robot controller.

Procedure

1. Click on the **Deploy…** button in the menu bar. The **WorkVisual Project Deployment** window is opened.
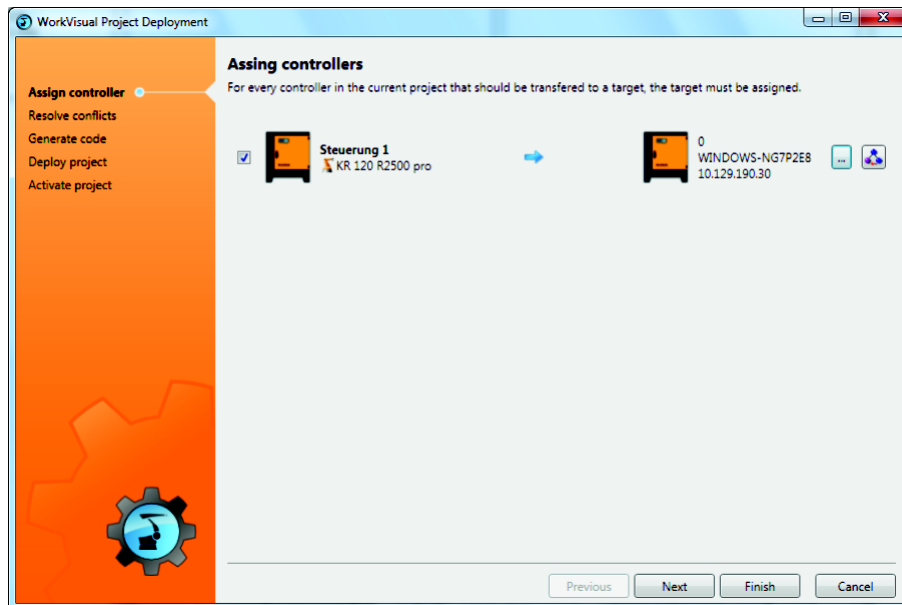
Fig. 13-5: Overview

2. Click on **Next**. Program generation begins.

3. Click on **Next**. The project is transferred.

> **i** If a project has been transferred containing an option package that was inserted into the project without a window showing the changes being displayed:
> Do not activate the project, i.e. do not continue with the next step! Observe the note about projects with option packages at the beginning of this section!

4. Click on **Next**.

> ⚠ **WARNING** In the operating modes AUT and AUT EXT, the project is activated without any request for confirmation if there are only program changes.

5. Only in operating modes T1 and T2: The KUKA smartHMI displays the request for confirmation *Do you want to activate the project […]?*. In addition, a message is displayed as to whether the activation would overwrite a project, and if so, which.

   If no relevant project will be overwritten: Confirm with **Yes** within 30 minutes.

6. An overview is displayed of the changes which will be made in comparison to the project that is still active on the robot controller. The check box **Details** can be used to display details about the changes.

> ⚠ **WARNING** If changes are listed in the overview under the heading **Safety-relevant communication parameters**, this means that the behavior of the Emergency Stop and "Operator safety" signal may have changed compared with the previous project.
> After activation of the project, the Emergency Stop and the "Operator safety" signal must be checked for safe functioning. If the project is activated on several robot controllers, this check must be carried out for every robot controller. Failure to carry out this check may result in death, injuries or damage to property.

7. The overview displays the request for confirmation *Do you want to continue?*. Confirm with **Yes**. The project is activated on the robot controller. A confirmation is displayed in WorkVisual.

Fig. 13-6: Confirmation in WorkVisual

8. Close the **WorkVisual Project Deployment** window by selecting **Finish**.
9. If the request for confirmation on the robot controller is not answered within 30 minutes, the project is still transferred, but is not activated on the robot controller. The project can then be activated separately.

    (>>> 13.5 "Activating a project" Page 142)

⚠ **WARNING**   After activation of a project on the robot controller, the safety configuration must be checked there! If this is not done, the robot will possibly be operated with incorrect data. Death, injuries or damage to property may result.
 (>>> 13.6 "Checking the safety configuration of the robot controller" Page 144)

⚠ **WARNING**   If the activation of a project fails, an error message is displayed in WorkVisual. In this case, one of the following measures must be carried out:

■ Rectify the fault and restart the activation by clicking on the 🔀 button.
■ Or activate a different project.
■ Or: Reboot the robot controller with a cold restart.

13.5   Activating a project

■ A project can be activated on the robot controller from within WorkVisual.
    (>>> 13.5.1 "Activating a project (in WorkVisual)" Page 142)
■ A project can be activated directly on the robot controller.

ℹ   Information about activation on the robot controller can be found in the **Operating and Programming Instructions for System Integrators** for the KUKA System Software.

13.5.1   Activating a project (in WorkVisual)

Precondition   ■ Network connection to the real robot controller
■ The real robot controller and the KUKA smartHMI are running.

■ The user group "Expert" or higher is selected on the real robot controller.

Restriction: If the activation would cause changes in the area **Safety-relevant communication parameters**, the user group "Safety recovery" or higher must be selected.

■ If the operating mode AUT or AUT EXT is selected on the real robot controller: The project can only be activated if this affects only KRL programs. If the project contains settings that would cause other changes, it cannot be activated.

> **i** If one of the options KUKA.SafeOperation or KUKA.SafeRangeMonitoring is installed on the robot controller, different user groups may apply. Information can be found in the documentation for these options.

> **i** If a project has been transferred containing an option package that was inserted into the project without a window showing the changes being displayed:
> Do not activate the project! For such projects, the procedure differs from the usual procedure for transfer and activation. Further information can be found in the section on project deployment.
> (>>> 13.4 "Transferring the project to the robot controller" Page 140)

Procedure

1. Select the menu sequence: **File** > **Browse for project**. The **Project Explorer** is opened. On the left, the **Browse** tab is selected.

2. In the **Available cells** area, expand the node of the desired cell. All the robot controllers of this cell are displayed.

3. Expand the node of the desired robot controller. All projects are displayed. The active project is indicated by a small green arrow.

4. Select the desired project and click on the **Activate project** button. The **Deploy Project** window is opened.

5. Click on **Next**.

> ⚠ **WARNING** In the operating modes AUT and AUT EXT, the project is activated without any request for confirmation if there are only program changes.

6. Only in operating modes T1 and T2: The KUKA smartHMI displays the request for confirmation *Do you want to activate the project […]?*. In addition, a message is displayed as to whether the activation would overwrite a project, and if so, which.

If no relevant project will be overwritten: Confirm with **Yes** within 30 minutes.

7. On the KUKA smartHMI, an overview is displayed of the changes which will be made in comparison to the project that is still active on the robot controller. The check box **Details** can be used to display details about the changes.

> ⚠ **WARNING** If changes are listed in the overview under the heading **Safety-relevant communication parameters**, this means that the behavior of the Emergency Stop and "Operator safety" signal may have changed compared with the previous project.
> After activation of the project, the Emergency Stop and the "Operator safety" signal must be checked for safe functioning. If the project is activated on several robot controllers, this check must be carried out for every robot controller. Failure to carry out this check may result in death, injuries or damage to property.

8.   The overview displays the request for confirmation *Do you want to continue?*. Confirm with **Yes**. The project is activated on the robot controller. A confirmation is displayed in WorkVisual.
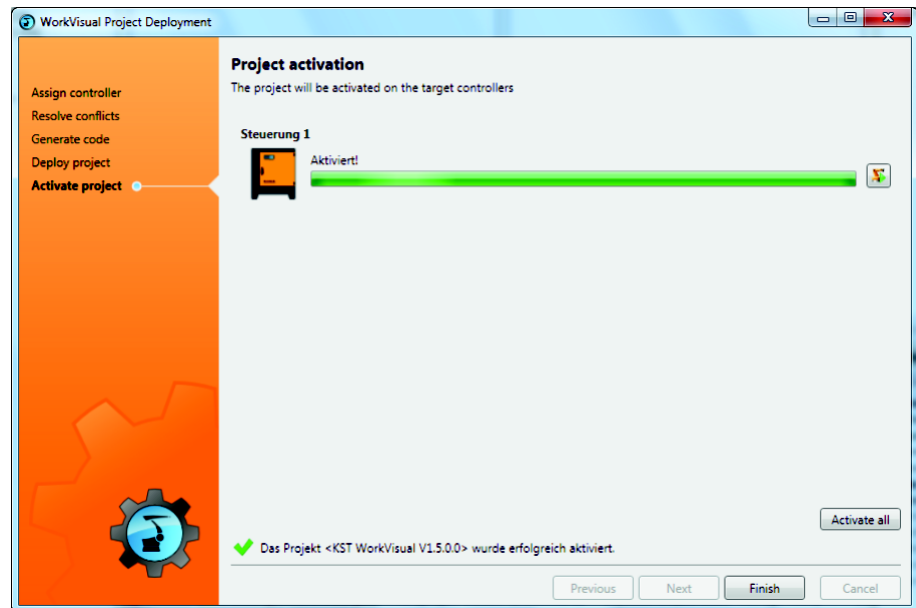
9.   In WorkVisual, close the **Project deployment** window by selecting **Exit**.

10.   Click on **Refresh** in the **Project Explorer**. The active project is now indicated by a small green arrow. (The small green arrow disappears from the project that was active before.)

> ⚠ **WARNING**   After activation of a project on the robot controller, the safety configuration must be checked there! If this is not done, the robot will possibly be operated with incorrect data. Death, injuries or damage to property may result.

> ⚠ **WARNING**   If the activation of a project fails, an error message is displayed in WorkVisual. In this case, one of the following measures must be carried out:
> - Either: Activate a project again (the same one or a different one).
> - Or: Reboot the robot controller with a cold restart.

13.6   Checking the safety configuration of the robot controller

Description

The safety configuration of the robot controller must be checked in the following cases:

- After activation of a WorkVisual project on the robot controller
- Generally after changes to the machine data (independent of WorkVisual).

> ⚠ **WARNING**   If the safety configuration is not checked and updated where necessary, it may contain incorrect data. Death, injuries or damage to property may result.

> ℹ   Information about checking the safety configuration and the safe axis monitoring functions is contained in the Operating and Programming Instructions for System Integrators.

13.7   Loading the project from the robot controller

Description

On every robot controller to which a network connection is established, a project can be selected and loaded in WorkVisual. This is also possible if this project is not yet present on this PC.

The project is saved in the directory: …\WorkVisual Projects\Downloaded Projects.

Precondition

- Network connection to the real robot controller

Procedure

1.   Select the menu sequence: **File** > **Browse for project**. The **Project Explorer** is opened. On the left, the **Search** tab is selected.

2.   In the **Available cells** area, expand the node of the desired cell. All the robot controllers of this cell are displayed.

3.   Expand the node of the desired robot controller. All projects are displayed.

4.   Select the desired project and click on **Open**. The project is opened in WorkVisual.

13.8 Comparing projects (and accepting differences)

Description   A project in WorkVisual can be compared with another project. This can be a project on a robot controller or a locally saved project. The differences are clearly listed. The user can decide for each individual difference whether to leave the state as in the current project or to transfer the state from the other project.

Precondition
- The project that is to be compared is open in WorkVisual.

If the other project to be compared is located on a robot controller:

- The real robot controller is running.
- Network connection to the real robot controller

Procedure
1. In WorkVisual, select the menu sequence **Extras** > **Compare projects**. The **Compare projects** window is opened.
2. Select the project with which the current WorkVisual project should be compared, e.g. the project of the same name on the real robot controller.
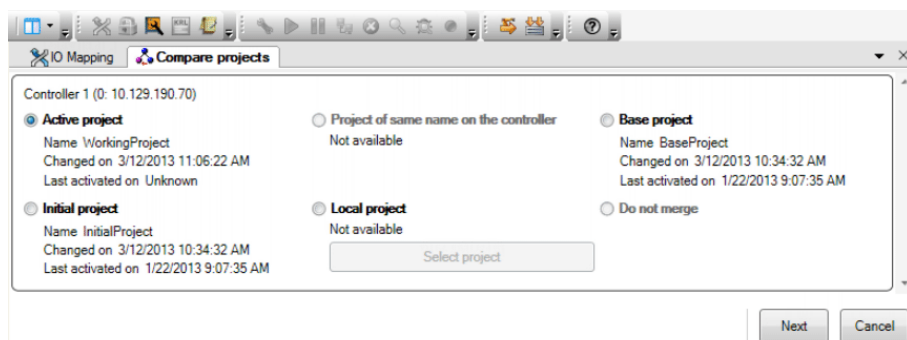

Fig. 13-7: Selecting a project for "Compare"

3. Click on **Next**. A progress bar is displayed. (If the project contains more than one controller, a bar is displayed for each one.)

    (>>> "Progress bars" Page 146)

4. When the progress bar is full and the message **Status: Ready for merge** is displayed: Click on **Show differences**. The differences between the projects are displayed in an overview.

    (>>> "Comparison" Page 146)

    If no differences were determined, this is indicated in the message window. Continue with step 8. After this, no further steps are necessary.

5. For each difference, select whether to retain the state of the current project or to adopt the state of the comparison project. This does not have to be done for all the differences at one go.

    If suitable, the default selection can also be accepted.

6. Press **Merge** to transfer the changes to WorkVisual.

7. Repeat steps 5 to 6 as required. This makes it possible to work through the different areas bit by bit.

    Once all the differences have been reconciled, the following message is displayed: **No further differences were detected.**

8. Close the **Compare projects** window.

9. Save the project.
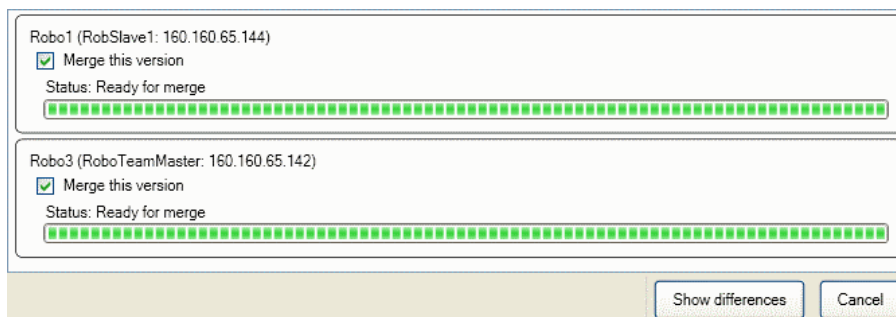
Progress bars

Fig. 13-8: Example: progress bar

This view shows all the robot controllers which are contained in the project. A separate bar is displayed for each of these robot controllers. For each bar, the real robot controller to which the project was last transferred is also specified. The check boxes are used to select for which robot controllers the comparison should be carried out.

If additional robot controllers were added or removed in WorkVisual after deployment, these robot controllers are also displayed here. They are marked as invalid, however, and cannot be selected.

Comparison

The differences between the projects are displayed in an overview. For each difference, the user can select which state to accept. The default setting is as follows:

■ For all elements that are present in the open project, the state of this project is selected.

■ For all elements that are not present in the open project, the state of the comparison project is selected.

> Exception: For projects with a VKRC 4 controller, the state of the comparison project is always selected for the long texts.
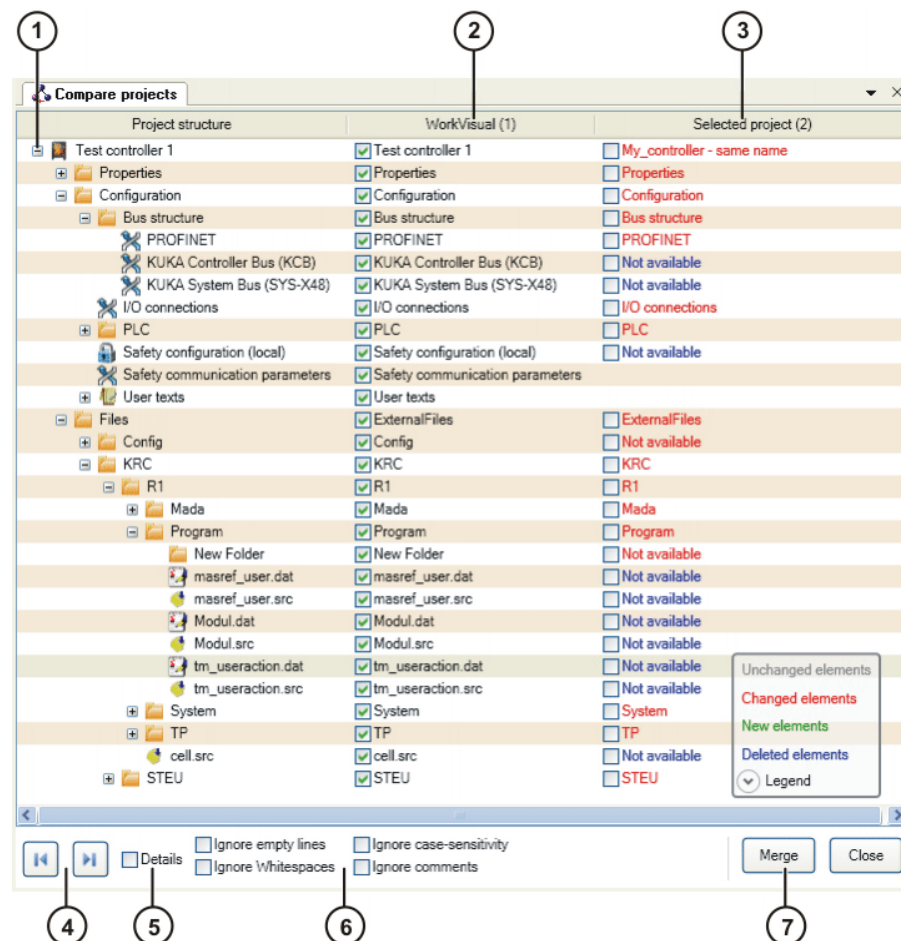
Fig. 13-9: Example: overview of differences

Item	Description
1	The node for the robot controller. The various project areas are represented by sub-nodes. The nodes can be expanded to display the comparisons.
	If several robot controllers are present, these are listed one after the other.
	■ In each line, place a check mark in the box for the value that should be transferred.
	■ A check mark next to **Not available** means that the element will not be transferred, or that it will be deleted from the project if already present.
	■ If a check box is activated for a node, the check boxes for all subordinate elements are also automatically activated.
	If a check box is deactivated for a node, the check boxes for all subordinated elements are also automatically deactivated.
	The subordinate elements can also be edited individually, however.
	■ A filled-in box means: at least one of the subordinate elements is selected, but not all.
2	State of the project that is open in WorkVisual
3	State of the comparison project
4	Back arrow: The focus in the display jumps to the previous difference.
	Forward arrow: The focus in the display jumps to the next difference.
	Collapsed nodes are automatically expanded.
5	TRUE: Detailed information is shown for the selected line in the overview.
6	Filter
7	Transfers the selected changes to the open project.

14 Diagnosis

14.1 Project analysis

14.1.1 Analyzing a project automatically for errors

Description   In WorkVisual, it is possible to analyze the current project continuously in the background. If configuration errors are detected, WorkVisual alerts the user. In addition, an automatic correction is offered for many errors.

An icon in the bottom right corner of the user interface indicates the status of the project analysis.

Icon	Color	Description
	Green	WorkVisual has not detected any errors.
	Yellow	WorkVisual has detected a discrepancy. The discrepancy will not prevent the project from running on a real robot controller. However, the discrepancy is probably not intended or desired by the user.
	Red	WorkVisual has detected an error. The project is unable to run on a real robot controller in this state. The error would come to light during code generation or, at the very latest, on the real robot controller.
	Gray	The analysis is deactivated.

> **i** The project analysis detects a wide range of errors and gives the user valuable support. A green icon cannot guarantee that the project is free from configuration errors, however.

Precondition   ■ Project analysis is activated.

Procedure   1. Depending on the configuration:
- The **WorkVisual project analysis** window opens automatically if the icon is red or yellow.
- Or: Click on the icon to open the **WorkVisual project analysis** window.
2. The window displays a brief description of the error. Often one or more possibilities for correction are displayed beneath the description.

Click on the desired correction proposal.
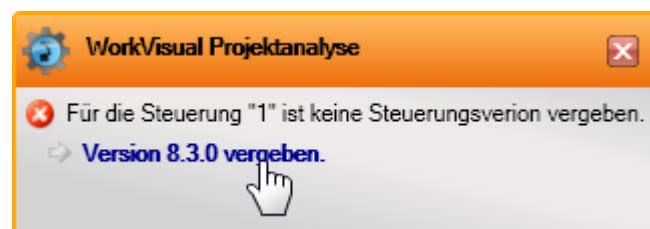

Fig. 14-1: Example: WorkVisual project analysis, with correction proposal

14.1.2 Configuring the project analysis function

Procedure   1. Select the menu sequence **Extras** > **Options**. The **Options** window is opened.

2. On the left of the window, select the **Project analysis** folder.

   The corresponding settings are now displayed on the right of the window.

3. Make the desired settings. Confirm with **OK**.

Description Setting options in the **Project analysis** folder:

Item	Description
Analysis activated	■ **Check box active**: The project is analyzed continuously. If errors or discrepancies are detected, these are displayed in the **WorkVisual project analysis** window. ■ **Without**: The project is not analyzed and no messages are displayed.
Automatic notification activated	■ **Check box active**: The **WorkVisual project analysis** window opens automatically whenever an error or discrepancy is detected. ■ **Without**: The **WorkVisual project analysis** window opens only when the icon is clicked.

14.2 Trace

Trace recordings are an important diagnostic tool during start-up of the industrial robot and during troubleshooting. They are also used for optimization of the machine data. The trace function can be used to record different variables with the program running, e.g. actual current, setpoint current, states of inputs and outputs, etc. The recording can then be displayed using the oscilloscope.

In WorkVisual it is possible to configure trace recordings and transfer them to the robot controller. The recording can also be started in WorkVisual. In addition, trace configurations can be imported from the robot controller to WorkVisual. The results of trace recordings can also be imported to WorkVisual. The oscilloscope function is also available here for display and evaluation.

14.2.1 Configuring and starting the trace recording

Description During configuration, the data to be recorded are specified. The robot controller saves the recording in the directory: C:\KRC\ROBOTER\TRACE.

Precondition ■ **Programming and diagnosis** workspace

Procedure 1. Select the menu sequence **Editors** > **Trace configuration**. The **Trace configuration** window opens.

2. Select a configuration or create a new configuration on the **General** tab. Edit the configuration if required.

3. In the **Cell view** window, select the robot controllers to which the configuration is to be transferred.

4. On the **General** tab, click on the **Save configuration on controller** button.

5. Respond to the request for confirmation asking whether the configuration should be activated by pressing **Yes**.

6. Click on the **Start trace** button to start the recording. The recording is started in accordance with the defined trigger.

   Or: Click on **Trigger**. The recording starts immediately.

   The **State** box jumps from #T_END to either #T_WAIT or #TRIGGERED.

7. The recording is ended when the **State** box displays the value #T_END again.

14.2.2   Importing a trace configuration

Description

Trace configurations can be imported. They are then available under **local** in the **Source** box of the **Trace configuration** window.

Procedure

1. The import/export function can be accessed in one of the following ways:
   - Select the menu sequence **Editors** > **Trace configuration**. The **Trace configuration** window is opened.

     On the **General** tab, click on the **Import/export of trace configurations** button.

   Or:
   - Select the menu sequence **File** > **Import / Export**. A window opens.

     Select **Import/export trace configurations** and click on **Next**.
2. Select the **Import** option.
3. If the desired directory is not displayed in the **Source directory** box:

   Click on **Browse** and navigate to the directory where the configuration is located. Select the directory and confirm selection with **OK**.

   The configurations located in the directory are displayed.
4. Specify whether existing data are to be overwritten.
5. Click on **Finish**.
6. The data are imported. If the import was successful, this is indicated by a message in the window. Close the window.

14.2.3   Exporting a trace configuration

Procedure

1. The import/export function can be accessed in one of the following ways:
   - Select the menu sequence **Editors** > **Trace configuration**. The **Trace configuration** window is opened.

     On the **General** tab, click on the **Import/export of trace configurations** button.

   Or:
   - Select the menu sequence **File** > **Import / Export**. A window opens.

     Select **Import/export trace configurations** and click on **Next**.
2. Select the **Export** option. All configurations stored locally are displayed.
3. If the desired directory is not displayed in the **Target directory** box:

   Click on **Browse** and navigate to the desired directory. Select the directory and confirm selection with **OK**.
4. Specify whether existing data are to be overwritten.
5. Click on **Finish**.
6. The data are exported. If the export was successful, this is indicated by a message in the window. Close the window.

14.2.4 "Trace configuration" window
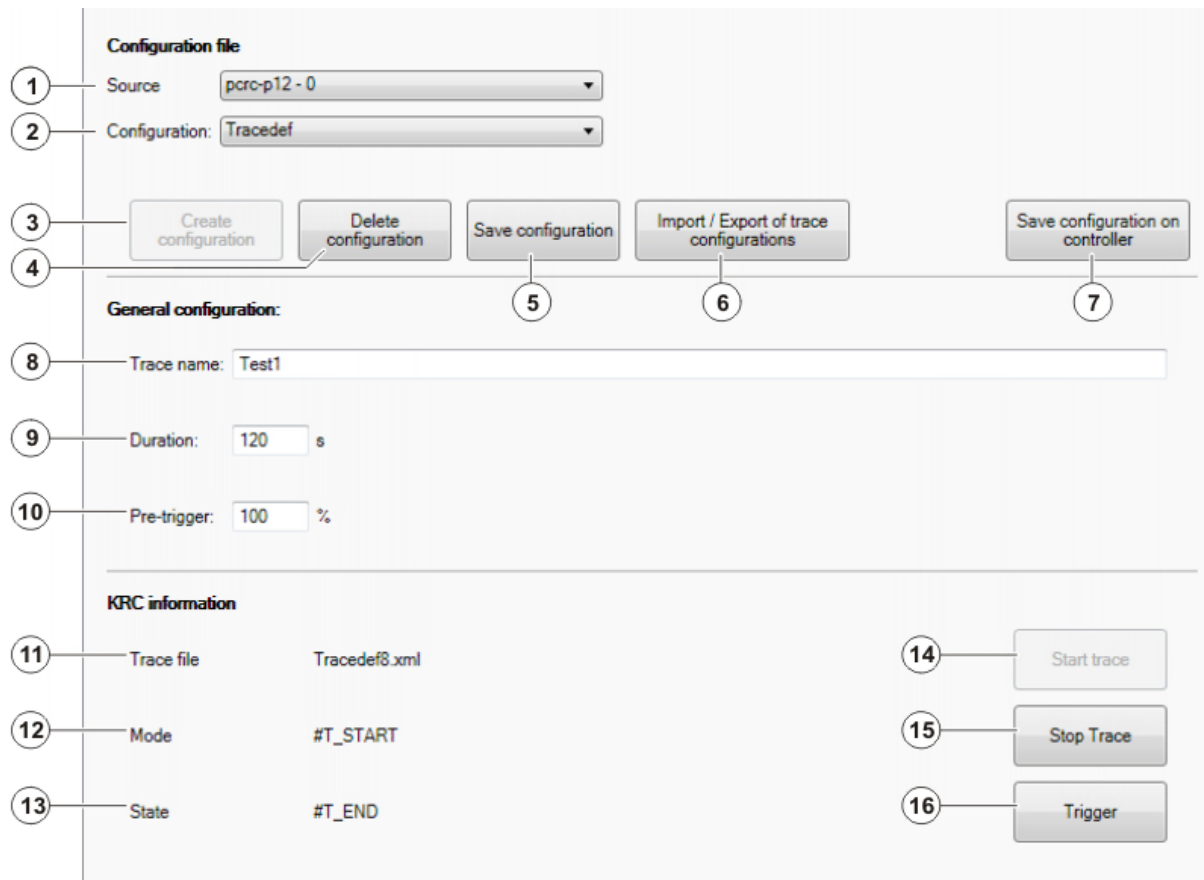
14.2.4.1 "General" tab

Fig. 14-2: "General" tab

Item	Description
1	■ **local**: The predefined and the locally saved configurations are available for selection in the **Configuration** box. ■ [*Robot controller*]: All the configurations saved on this robot controller are available for selection in the **Configuration** box. (In addition to those under **local**.) Robot controllers are only displayed in the **Source** box if they are selected in the **Cell view** window.
2	A configuration can be selected here. The configuration can be edited on the tabs and then saved locally or on a robot controller.
3	Opens a window in which a name for a new configuration can be entered. An existing local configuration can be selected as a template for the new configuration. On confirming the entries with **OK**, the new configuration is inserted in the list under **local**. This button is only displayed if the entry **local** is selected in the **Source** box.
4	Deletes the configuration displayed in the **Configuration** box.
5	Saves locally the configuration displayed in the **Configuration** box. The settings on the tabs are accepted.
6	Opens a window for the import/export of trace configurations.
7	Activates the configuration displayed in the **Configuration** box on the robot controllers selected in the **Cell view** window. If the request for confirmation is answered with **No**, the configuration is still saved on the robot controller, but is not activated there.

Item	Description
8	Name for the recording. The name can be changed. The robot controller adds extensions to the end of the name, indicating what data have been recorded.
9	Duration of the recording. Only whole numbers can be entered. Maximum value: 9999 s
10	The position of the time phase displayed in the recording relative to the trigger. The % value refers to the duration of the recording. Examples: ■ **0%**: The displayed time phase starts at the trigger. ■ **30%**: 30% of the displayed time phase comes before the trigger, 70% after the trigger. ■ **100%**: The displayed time phase ends at the trigger.
	All of the following elements are only displayed if a robot controller is selected in the **Source** box.
11	Trace configuration that is currently active on the robot controller.
12	■ **#T_START**: Recording is running. ■ **#T_STOP**: Recording is not running.
13	State of the recording ■ **#T_WAIT**: The recording is started and is waiting for the trigger. ■ **#TRIGGERED**: The recording continues for the time defined by the trace length and trigger. ■ **#T_END**: No recording is running.
14	Starts the recording with the configuration displayed under **Trace file**. This button is only displayed if no recording has been started yet.
15	Stops the recording. This button is only displayed if a recording has been started.
16	Starts the recording. This button is only displayed if a recording has already been started. Strictly speaking, data are recorded as soon as the **Start trace** button is pressed. The trigger merely controls which time phase of the recording is then displayed in the trace files.

14.2.4.2 "Trigger" tab

Triggers can be selected here. Triggers control when data is recorded.

Strictly speaking, data are recorded as soon as the **Start trace** button is pressed. The trigger merely controls which time phase of the recording is then displayed in the trace files.
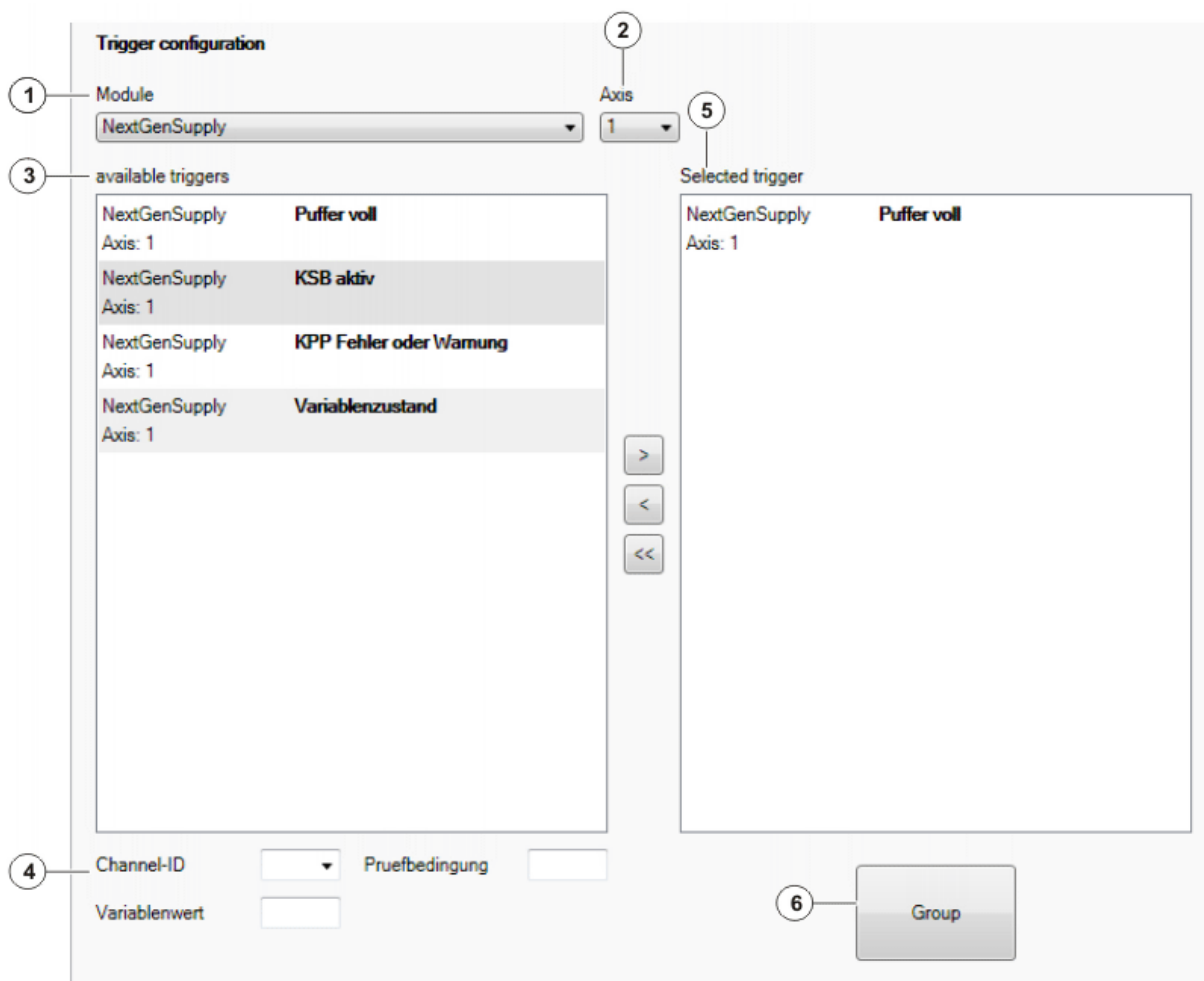
Fig. 14-3: "Trigger" tab

Item	Description
1	A module can be selected here. The modules contain numerous predefined triggers.
2	This box is only displayed if the selected module refers to the robot axes. It is possible to select the axes to which the triggers are to refer.
3	All the triggers for this module are displayed here.
	RIGHT ARROW copies triggers selected here to the **Selected trigger** box. (Alternatively: double-click on a trigger.)
4	Depending on the entry selected under **available triggers**, filters are available here for this entry.
5	Here the triggers are inserted that are to be used for the current configuration.
	LEFT ARROW removes the triggers selected here. (Alternatively: double-click on a trigger.)
	DOUBLE LEFT ARROW clears this box.
	Group combines all entries of the same type into a single entry. This gives greater clarity to the display. It has no effect on the recording.
	Ungroup undoes the grouping.

14.2.4.3 "I/O" tab

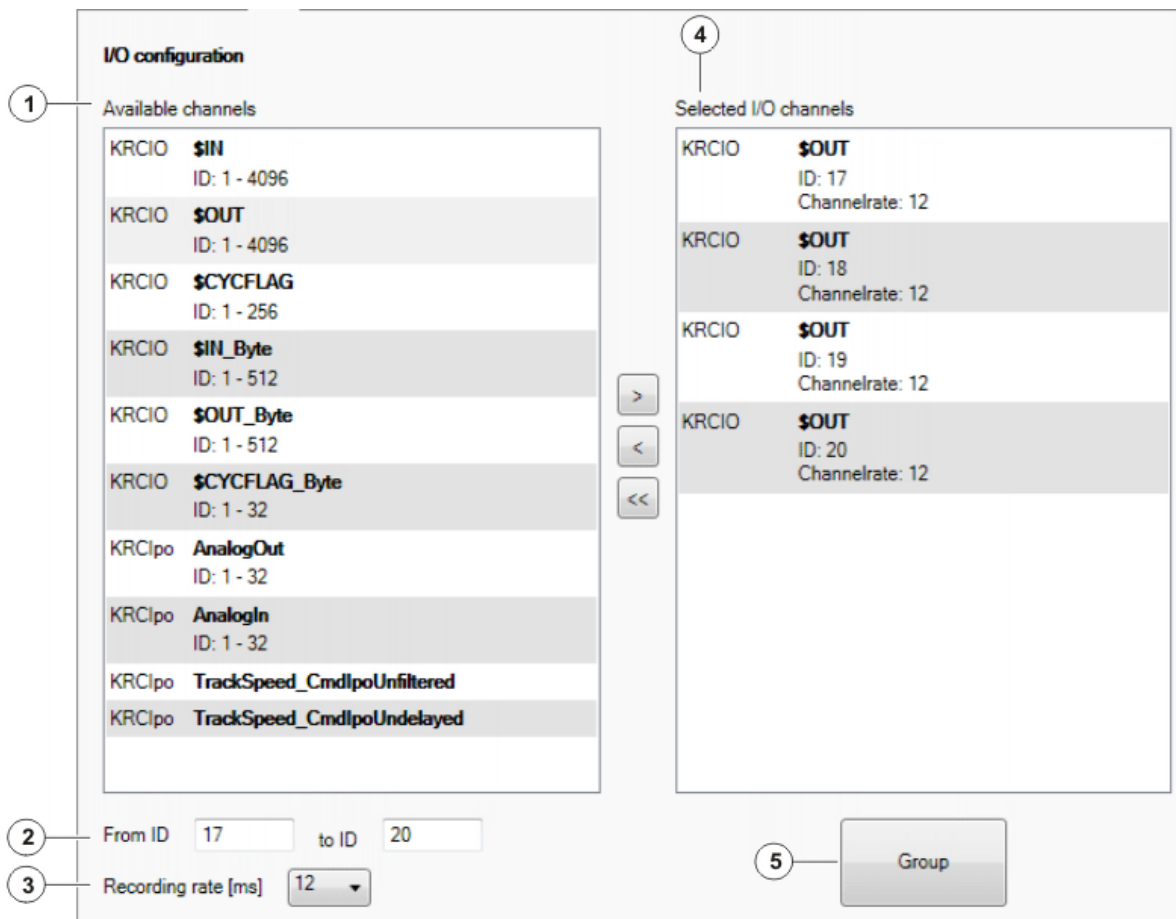Here you can select which inputs or outputs are to be recorded.

Fig. 14-4: "I/O" tab

Item	Description
1	All the available inputs/outputs are displayed here.
2	Here a number range can be specified from the entry selected under **Available channels**.
3	Select the desired recording rate.
4	Here the inputs/outputs are inserted that are to be recorded with the current configuration. RIGHT ARROW transfers all the inputs/outputs selected via **Available channels** and **From ID […] to ID […]** to this box. LEFT ARROW removes the inputs/outputs selected here. (Alternative to these arrow keys: double-click on a channel.) DOUBLE LEFT ARROW clears this box.
5	**Group** combines all entries of the same type into a single entry. This gives greater clarity to the display. It has no effect on the recording. **Ungroup** undoes the grouping.

14.2.4.4 "Configuration" tab

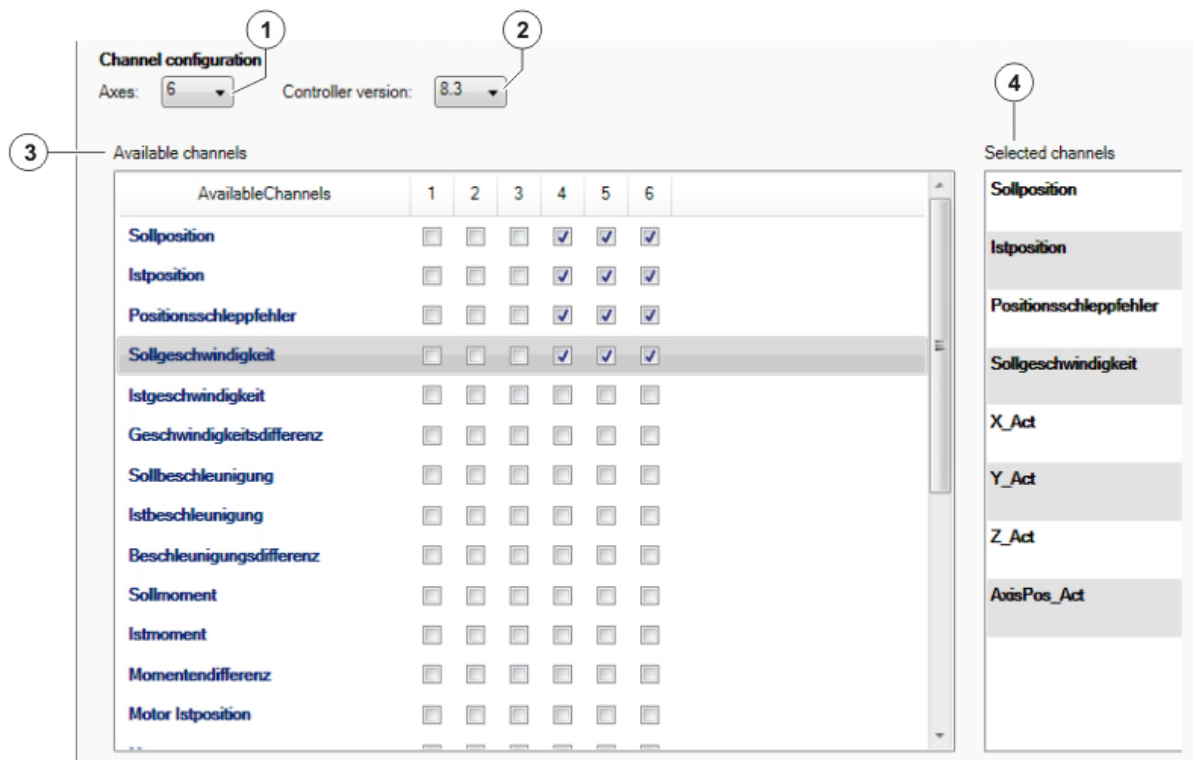The settings on the **Extended configuration** tab are also displayed on the **Configuration** tab, and vice versa.

Fig. 14-5: "Configuration" tab

Item	Description
1	It is possible to select here how many axes are displayed in the **Available channels** display.
2	The version of the system software to which the entries on this tab refer must be selected here. **Note:** First select the correct version here, then make the other settings on this tab! Is the version is changed, any channels that have been selected are discarded.
3	Here you can select the channels that are to be recorded. ■ To select a channel for individual axes, check the box for the relevant axes. ■ To check or uncheck all the boxes in one row, double-click on the row. Channels that are not axis-specific have only one check box.
4	All the selected channels are displayed here. **Note:** It is possible that channels may be displayed here which cannot be selected on the **Configuration** tab. This is the case if these channels have been selected on the **Extended configuration** tab.

14.2.4.5 "Extended configuration" tab

The **Extended configuration** tab contains the same modules as the **Configuration** tab, but with more channels and more detailed selection options. **Extended configuration** is particularly suitable for expert users.

The settings on the **Extended configuration** tab are also displayed on the **Configuration** tab, and vice versa.
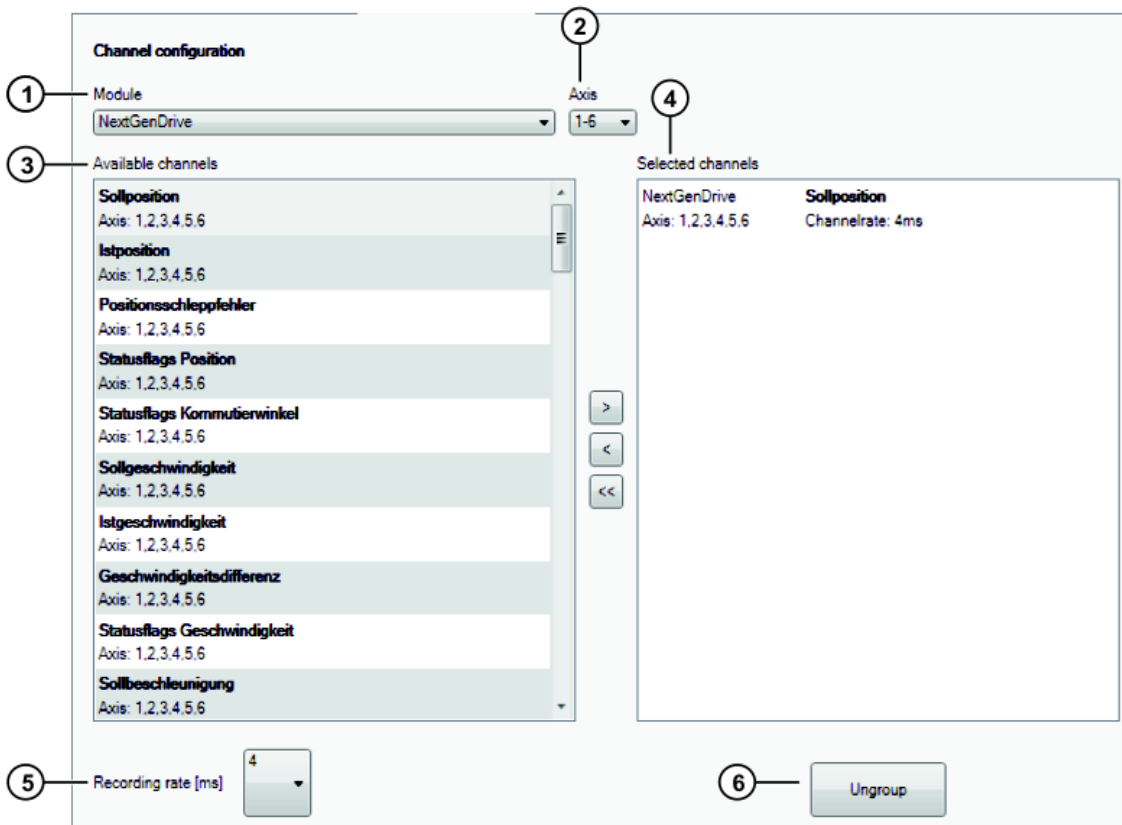
Fig. 14-6: "Extended configuration" tab

Item	Description
1	A module can be selected here. The modules contain different channels.
2	This box is only displayed if the selected module refers to the robot axes. It is possible to select the axes to which the channels are to refer.
3	All the channels for the selected module are displayed here.
4	Here the channels are inserted that are to be recorded with the current configuration. RIGHT ARROW transfers all the entries selected under **Available channels** to this box. LEFT ARROW removes the entries selected here. (Alternative to these arrow keys: double-click on an entry.) DOUBLE LEFT ARROW clears this box.
5	Select the desired recording rate.
6	**Group** combines all entries of the same type into a single entry. This gives greater clarity to the display. It has no effect on the recording. **Ungroup** undoes the grouping.

14.2.5 Importing a trace recording

Description To be able to display a trace recording in WorkVisual, it must first be imported.

Precondition ■ The recording was created with KSS or VSS 8.1 or higher, or with 5.4.

Procedure
1. Select the menu sequence **File** > **Import / Export**. A window opens.
2. Select **Import trace results** and click on **Next**.
3. Click on **Browse** and navigate to the directory where the results are located. Select the directory and confirm selection with **OK**.
   All trace files located in the directory are displayed.
4. Select the traces to be imported.

5. Specify whether existing data are to be overwritten.

6. Select the appropriate entry in the **Format** box.

7. Click on **Finish**.

8. The data are imported. If the import was successful, this is indicated by a message in the window. Close the window.

14.2.6 Displaying a trace recording

Precondition
- **Programming and diagnosis** workspace
- The recording has been imported to WorkVisual.

Procedure
1. Select the menu sequence **Editors** > **Trace Analysis (Oscilloscope)**. The **Trace Analysis (Oscilloscope)** window opens.
2. On the **Channels** tab, select a recording.

   (>>> 14.2.7.1 ""Channels" tab" Page 158)
3. Select the channels to be displayed.
4. The channels are displayed on the **Oscilloscope** tab. Adapt the display, if necessary. (E.g. zoom or change the colors of the traces.)

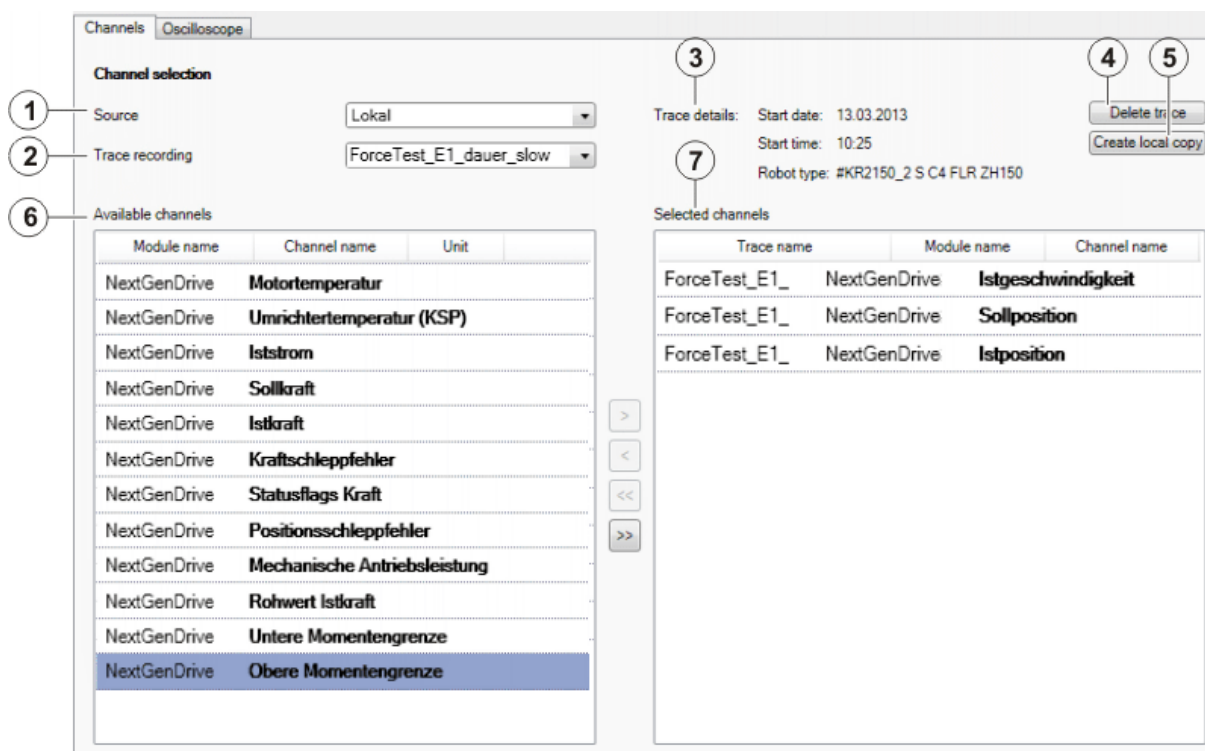### 14.2.7 "Trace Analysis" window

14.2.7.1 "Channels" tab

Fig. 14-7: "Channels" tab

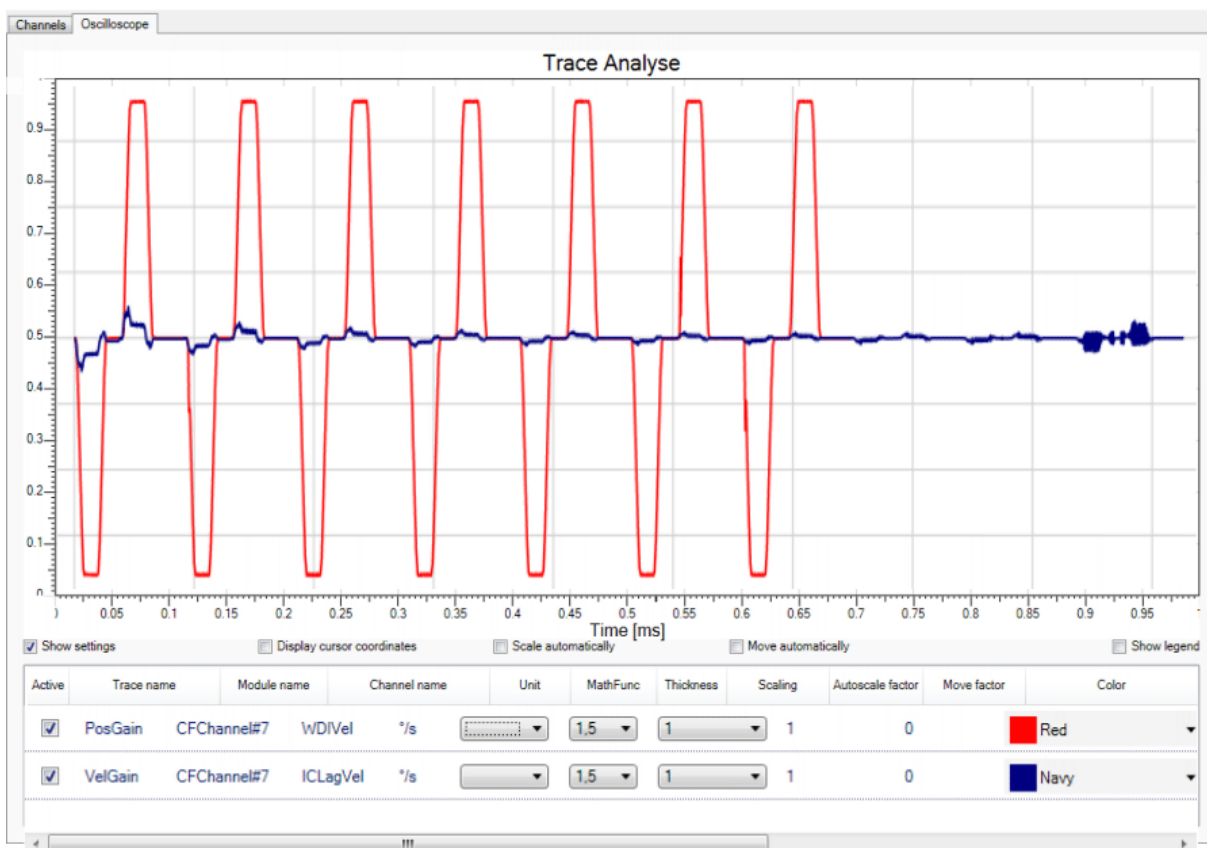| Item	Description
1	■ **Local**: All the locally saved recordings are available for selection in the **Trace recording** box.
	■ [*Robot controller*]: All the recordings saved on this robot controller are available for selection in the **Trace recording** box. (In addition to those under **local**.)
	Robot controllers are only displayed in the **Source** box if they are selected in the **Cell view** window.
2	A recording can be selected here.
3	Detailed information regarding the selected recording is displayed here.
4	Only active if the entry **Local** is selected in the **Source** box:
	Deletes the recording selected in the **Trace recording** box.
5	Only active if a robot controller is selected in the **Source** box:
	Creates a local copy of the selected recording.
6	All the channels contained in the selected recording are displayed here.
7	Here the channels are inserted that are to be displayed in the oscilloscope. It is possible to insert entries from different recordings in this box.
	RIGHT ARROW moves all the entries selected under **Available channels** to this box. LEFT ARROW removes the entries selected here. (Alternative to these arrow keys: double-click on the entry.)
	DOUBLE LEFT ARROW clears this box.

14.2.7.2 "Oscilloscope" tab

Fig. 14-8: "Oscilloscope" tab

Check box	Description
Show settings	Activated: The columns **Active**, **Trace name**, etc., are displayed.
Display cursor coordinates	Activated: The X and Y coordinates of the mouse pointer position are displayed in the diagram.
Scale automatically	Activated: The curves are adapted to one another in size so that they are easier to compare visually. The proportions between the curves then usually no longer correspond to reality. The current factor is displayed under **Autoscale factor**, however.
Move automatically	Activated: The midpoints of the curves are aligned. This option makes it possible to compare curves which lie far apart on the Y axis. The Y values then usually no longer correspond to reality. The current factor is displayed under **Move factor**, however.
Show legend	Activated: The diagram displays which channel name belongs to which trace color.

Column	Description
Active	Activated: The curve is displayed in the oscilloscope. Deactivated: The curve is displayed in the oscilloscope.
Trace name	Name of recording
Module name	Name of the module
Channel name	Name of channel
Unit	Unit for the Y axis of the oscilloscope display (can be different for each curve)
MathFunc	Mathematical functions that can be applied to the curve. The graph for the function is displayed in a similar color to that of the curve.
Thickness	Line thickness of the curve (unit: point)
Scaling	This selection box allows the amplitude to be increased or decreased in steps. In this way, it is also possible to make curves more visible which only have a low amplitude or which are hidden by other curves.
Autoscale factor	Factor resulting from the option **Scale automatically**.
Move factor	Factor resulting from the option **Move automatically**.
Color	Color of the curve
Points	Activated: The motion blocks of the robot are displayed. Start and end are shown for each block.
Values	Activated: The individual values which make up the curve are displayed as points.
RMS	Activated: The RMS value is shown. **Note:** The RMS value refers to the phase of the recording represented in the oscilloscope. In electrical engineering, the RMS value is the root-mean-square value of a signal that changes over time. RMS = Root Mean Square
Stairs	Activated: The curve runs horizontally on the X plane from one value until it reaches the Y value of the next value. From there it runs vertically up to this Y value. Deactivated: The curve takes the shortest path from one value to the next.
Offset X axis	Offsets this curve on the X axis by the specified value.
Offset Y axis	Offsets this curve on the Y axis by the specified value.

Column	Description
Min.	The minimum value of the curve, with reference to the currently visible segment
Max.	The maximum value of the curve, with reference to the currently visible segment

14.2.8 Panning and zooming the oscilloscope display

Procedure

Panning:

1. Click in the display and hold down the mouse button.
2. Drag with the mouse. The display moves with the mouse.

Zooming:

1. Click into the display.
2. Scroll with the mouse wheel.

   Scroll down: Zoom gets smaller. Scroll up: Zoom gets larger.

Enlarging a section:

1. Hold down the SHIFT key.
2. Click in the display and hold down the mouse button.
3. Move the mouse over the desired section. A gray rectangle is displayed. The size can be changed by moving the mouse. (The aspect ratio cannot be changed.)
4. Release the mouse button. The contents of the gray rectangle are displayed in enlarged form.

In this procedure, the aspect ratio of the selected section is adapted to the oscilloscope display:

1. Hold down the CTRL key.
2. Click in the display and hold down the mouse button.
3. Move the mouse over the desired section. A gray rectangle is displayed. The size and the aspect ratio can be changed by moving the mouse.
4. Release the mouse button. The contents of the gray rectangle are displayed in enlarged form.

Restoring the default view:

1. Right-click in the display.
2. Select **Fit to view** from the context menu.

14.2.9 Creating a screenshot of the oscilloscope display

Procedure

Creating a screenshot in the clipboard:

1. Right-click in the display.
2. Select **Copy screenshot** from the context menu.

Creating and saving the screenshot:

1. Right-click in the display.
2. Select **Save screenshot** from the context menu. A window opens in which the target directory can be selected. The screenshot is saved there as a PNG file.

14.3   Recording network traffic

> This functionality is only available for robot controllers with version 8.3.

Description

WorkVisual can record the communication data of the Ethernet-based interfaces of the robot controller, e.g. PROFINET, EtherCAT and EtherNet/IP.

WorkVisual saves the recording in a PCAP file. The default directory is C:\User*Username*\My Documents. The directory and file name may be changed.

PCAP files can be displayed using software for the analysis of network communication connections ("sniffer" software). Sniffer software is not included in the scope of supply of WorkVisual.

Precondition

- Network connection to the real robot controller
- The active project has been loaded from the real robot controller.
- The robot controller has been set as the active controller in WorkVisual.

Procedure

1. If desired: in the bus structure of the project, select the element whose interface data are to be recorded. The interface is then automatically preselected later.
2. Click on the **Perform network capture...** button. The **Select the network interface** window is opened.

    (>>> Fig. 14-9)
3. Select the desired interface, if not already preselected.
4. If required: select the filter criteria.
5. Click on **Next >**.
6. To start recording, click on **Start**.

    A progress bar and a counter display the quantity of data recorded. The memory capacity is limited to 5 MB. If more data are recorded, the ring buffer is activated, i.e. the oldest data is progressively discarded and replaced by the newest incoming data.

    - As the memory fills for the first time, the progress bar grows to visualize how full the memory is.
    - When the ring buffer is activated, the text **Ring buffer active** is displayed. A green light now moves along the progress bar.
7. To stop recording, click on **Stop**.

    If required, recording can be started again via **Restart**. The existing data are then discarded.
8. To save the recording, click on **Next >** after stopping. The target directory and file name are displayed. The directory and file name may be changed.
9. Click on **Next >**. The recording is saved and the text **Import successful** is displayed.
10. Click on **Close**.

**Select the
network interface**

Fig. 14-9: "Select the network interface" window

Box	Description
Network interface	Select the interface whose communication data are to be recorded.
Ethertype	Here the data to be recorded can be limited to a particular type. If no limit is desired, select **All**.
Protocol	This box is only displayed if either **IPv4** or **IPv6** is selected under **Ethertype**. Here the data to be recorded can be limited to a particular protocol. If no limit is desired, select **All**.
MAC address	The data to be recorded can be limited to a particular MAC address. If no limit is desired, leave the box blank.

14.4 Displaying messages and system logs of the robot controller

> This functionality is only available for robot controllers with version 8.3.

Description The messages displayed in the message window on the smartHMI can also be displayed in WorkVisual. Messages generated by PROFINET or its field bus devices contain links in WorkVisual. These so-called "diagnostic links" guide the user to other areas of WorkVisual and help to trace the cause of the message.

In addition, the system logs of the robot controller can be displayed, i.e. the entries in the log memory. A search function and numerous filters are available here.

Precondition
- Network connection to the real robot controller
- The real robot controller and the KUKA smartHMI are running.
- **Programming and diagnosis** workspace

Procedure
1. In the **Cell view** window, select the desired robot controller by activating the check box. It is also possible to select more than one controller.
2. Select the menu sequence **Editors** > **Log view**. The **Log view** window opens. An entry is displayed for each robot controller selected.
3. Click on an entry to expand it. The following tabs are now displayed:
   - **MessageLogs**: Displays the messages of this robot controller.

■ **SystemLogs**: Displays the log entries of this robot controller.

14.4.1   MessageLogs tab
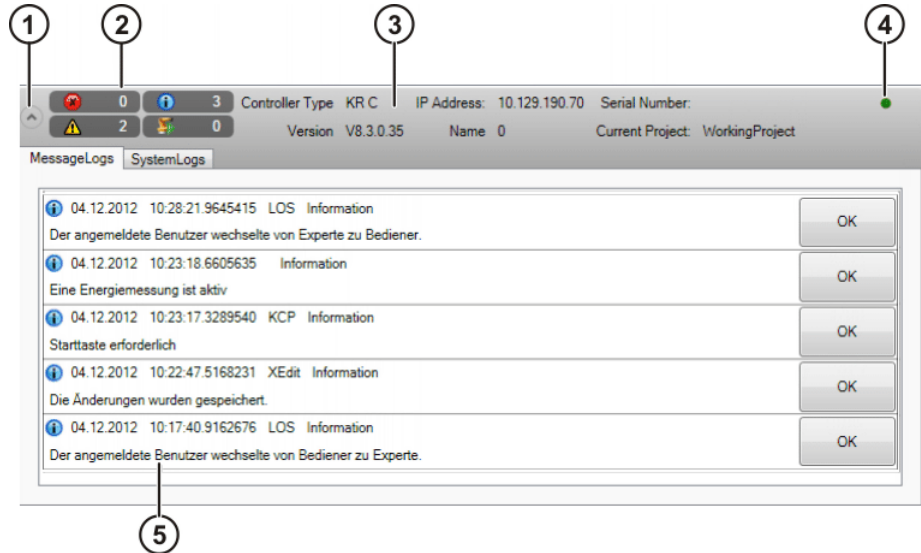
Fig. 14-10: Log view with "MessageLogs" tab

Item	Description
1	Click here (or on any part of the gray area) to expand or collapse the entry.
	The **MessageLogs** and **SystemLogs** tabs are visible when the entry is expanded.
2	Message counter
	The message counter indicates how many messages of each message type are active.
3	Information about the robot controller and the active project
	While the connection to the robot controller is being established, a lamp flashes next to the name of the active project. It disappears when the connection is established.
4	Status of the lamp:
	■ Green: A connection has been established to the real robot controller.
	■ Red: The connection to the real robot controller is terminated.
5	The messages displayed in the message window on the smartHMI are displayed here.
	■ If a message is acknowledged in the message window, it is also acknowledged here in **MessageLogs**.
	■ If a message is acknowledged here in **MessageLogs**, it is not acknowledged in the message window!
	Messages may contain diagnostic links.

Diagnostic links   Messages generated by PROFINET or its field bus devices contain links in WorkVisual. These so-called "diagnostic links" guide the user to other areas of WorkVisual and help to trace the cause of the message.

For the links **Online device diagnosis** and **Profinet device list**, the following applies:

■ If the active project has not yet been loaded from the robot controller, this is now done automatically. This is preceded by a request for confirmation.

■ If another project is open, this is closed. If it contains unsaved changes, a dialog is displayed, asking whether these changes should be saved.

Diagnostic link	Description
Diagnostic monitor	The link opens the diagnostic monitor. The device from which the message comes is automatically selected in the module overview.
Online device diagnosis	The link sets the device causing the error to "Connected", opens the **Diagnosis…** window and displays the **Device diagnosis** tab.
Profinet device list	The link sets the PROFINET node to "Connected", opens the **Device list and PROFINET names...** window and displays the **Available devices** tab.

> ℹ️ Information about the above-mentioned windows can be found in the documentation **KR C4 PROFINET**.

14.4.2 "SystemLogs" tab
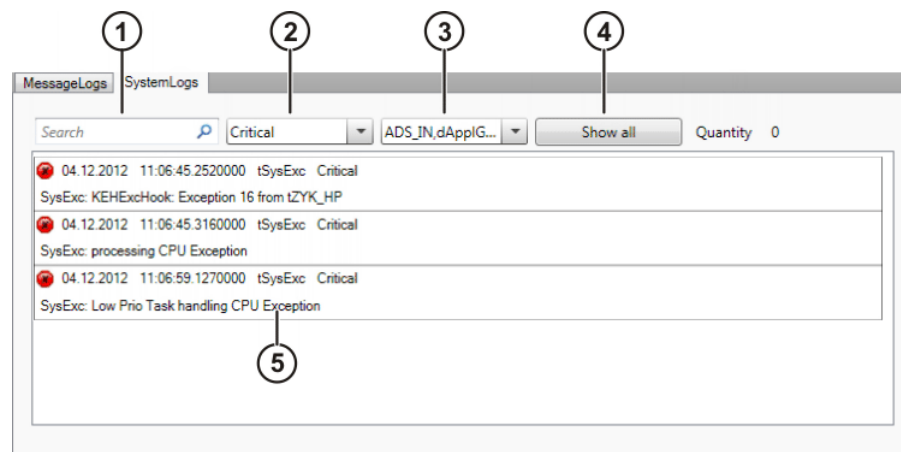
Fig. 14-11: SystemLogs

Item	Description
1	Here the system logs can be searched for one or more search terms. Upper and lower case are ignored. The order in which the search terms are entered in the search box is irrelevant. It is not necessary to search for complete words.
	Examples:
	■ *tick syst* also finds entries containing *system tick*.
	■ *tick tick* also finds entries in which *tick* occurs only once.
2	Filter: severity of the entry
	To set or remove a filter, expand the selection box and activate or deactivate the check boxes.
3	Filter: origin of the entry
	To set or remove a filter, expand the selection box and activate or deactivate the check boxes.

Item	Description
4	■ **Display none**: Removes all filters. ■ **Show all**: Sets all filters. This button has no effect on the search box.
5	System logs of the robot controller

14.5 Displaying diagnostic data about the robot controller

Description The diagnostic functionality makes it possible to display a wide range of diagnostic data concerning numerous software modules of a robot controller. The parameters displayed depend on the selected module. The display includes states, fault counters, message counters, etc.

Examples of modules:

■ **Kcp3 driver** (= drive for the smartPAD)
■ Network driver

"Lamps" indicate the status of the parameters, etc.:

■ **Green:** Status OK
■ **Yellow:** Status critical, could be faulty
■ **Red:** Error

Precondition ■ Network connection to the real robot controller
■ The real robot controller and the KUKA smartHMI are running.
■ **Programming and diagnosis** workspace

Procedure 1. In the **Cell view** window, select the desired robot controller by activating the check box. It is also possible to select more than one controller.
2. Select the menu sequence **Editors** > **Diagnostic monitor**. The **Diagnostic monitor** window opens.
3. An entry is displayed for each robot controller selected. Click on an entry to expand it. The following tabs are now displayed:
   ■ **Module view** (>>> 14.5.1 ""Module view" tab" Page 167)
   ■ **Signal diagram** (>>> 14.5.2 ""Signal diagram" tab" Page 169)
4. Select a module in the **Module view**. Diagnostic data are displayed for the selected module.

14.5.1 "Module view" tab

Fig. 14-12: Module view

Item	Description
1	Click here (or on any part of the gray area) to expand or collapse the entry. The **Module view** and **Signal diagram** tabs are visible when the entry is expanded.
2	Information about the robot controller and the active project While the connection to the robot controller is being established, a lamp flashes next to the name of the active project. It disappears when the connection is established.
3	This lamp indicates the status of the robot controller: ■ Red: When the status of at least one module is red. ■ Yellow: When the status of at least one module is yellow and no module is red. ■ Green: When the status of all modules is green.
4	Exports the chronological series of values to a LOG file. The values are sorted by time stamp. The time stamps start from the time at which the **Diagnostic monitor** was opened.
5	Module overview. Lamps indicate the status of the modules: ■ Red: When the status of at least one parameter is red. ■ Yellow: When the status of at least one parameter is yellow and no parameter is red. ■ Green: When the status of all parameters is green. **Note:** If the display refers to a robot controller with System Software 8.2, the module overview is not structured hierarchically.

Item	Description
6	Graphical representation of the topology for the following bus topologies: ■ Controller bus ■ KUKA Operator Panel Interface The lamp on a device is gray if the device is not connected to the real robot controller.
7	Diagnostic data about the selected module. Lamps indicate the status of the parameters: ■ Red: If the value lies outside of the range defined in the red box in the **Limit values** column. ■ Yellow: If the value lies outside of the range defined in the yellow box in the **Limit values** column. ■ Green: If the value lies within the range defined in the yellow box in the **Limit values** column.

Diagnostic data:

Column	Description
Name	Diagnosed parameter
Value	Current value of the diagnosed parameter
Extreme values	■ Upper value: Maximum diagnosed value ■ Lower value: Minimum diagnosed value The extreme values refer to the time since the diagnosis window was opened, unless the user clicks on the **Refresh** button (= green double arrow): determination of the extreme values then starts again.
Unit	If there is a unit associated with a parameter, this is displayed here. In some cases, the units can be changed (e.g. from seconds to milliseconds).
Limit values	This column partly contains default values. The values can be changed/specified by the user. Yellow box: ■ Upper value: If this value is exceeded, the parameter is marked yellow. ■ Lower value: If the current value falls below this value, the parameter is marked yellow. Red box: ■ Upper value: If this value is exceeded, the parameter is marked red. ■ Lower value: If the current value falls below this value, the parameter is marked red.
Color	Color of the curve on the **Signal diagram** tab

14.5.2 "Signal diagram" tab
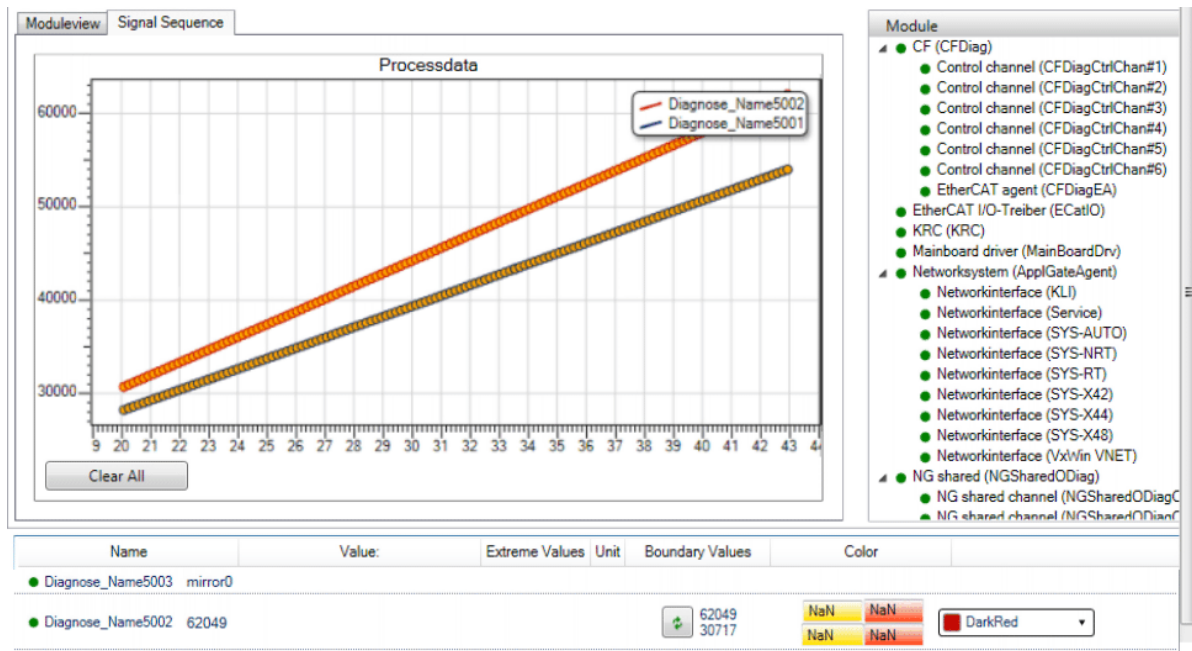
Fig. 14-13: Signal diagram

On the **Signal diagram** tab, the chronological sequence of values can be displayed graphically. Values to which a color has been assigned are displayed. If a different module is marked, the curves of the previous module are retained in the diagram. This makes it possible to compare graphs from different modules.

The **Clear all** button resets all color settings to **Transparent** and removes all curves from the diagram.

If the mouse pointer is moved over the diagram, the X and Y coordinates of the mouse pointer position are displayed. A context menu is available in the diagram, allowing the following actions:

- Adapting the size of the diagram so that the entire curve is displayed
- Creating a screenshot in the clipboard
- Saving the screenshot
- Opening the help function

 The help function contains useful information about zooming and keyboard shortcuts.

14.5.3 Diagnostic data for the transfer application module

Name	Description
Initialized	■ **Yes**: The transfer application is connected to all connected field buses. Data exchange is taking place. ■ **No**: No configured field bus device is present.
Number of transferred bits	Number of bits configured
Cycle time of the transfer data [ms]	Current cycle time of the transfer application
Processor load [%]	CPU load caused by the transfer application
Driver name	Name of the driver
Bus name	Name of the field bus

Name	Description
Bus status	■ **OK**: Status OK ■ **Error**: The field bus has an error
Bus connected	■ **Yes**: The connection to the field bus is established ■ **No**: No connection to the field bus

14.6 Displaying online system information

Precondition

■ **Programming and diagnosis** workspace

Procedure

1. In the **Cell view** window, select the desired robot controller by activating the check box. It is also possible to select more than one controller.
2. Select the menu sequence **Editors** > **System information editor**. The **Online system information** window opens. An entry is displayed for each robot controller selected.

Description
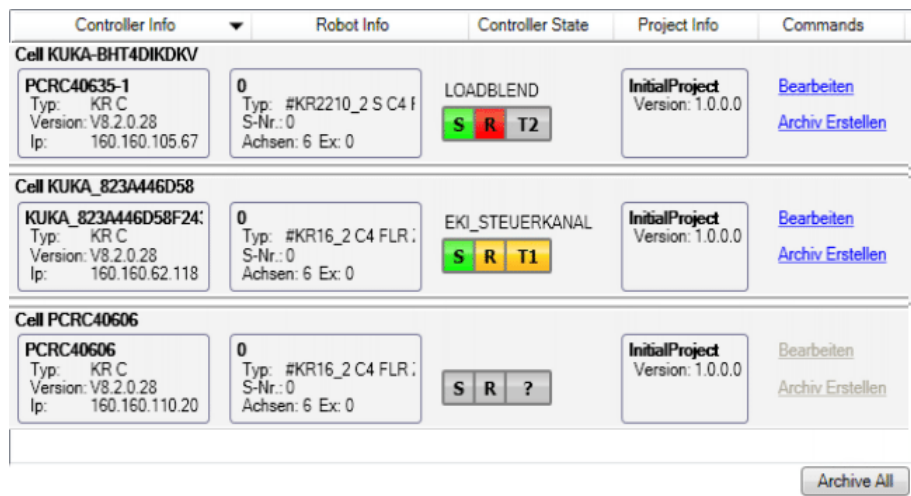
Fig. 14-14: "Online system information" window

Column	Description
Controller Info	Information about the robot controller is displayed here.
Robot info	Information about the robot is displayed here.
Controller status	Shows the status of the Submit interpreter and robot interpreter and the operating mode. The status displays correspond to the status displays on the KUKA smartHMI. Information about this can be found in the operating and programming instructions for the KUKA System Software (KSS).
Project Info	Information about the active project is displayed here.
commando	**Edit**: Opens the **Device properties** window. **Create archive**: Opens the **Generate archives** window. (The data for this robot controller can be archived.)

Button	Description
Archive all	**Create archive**: Opens the **Generate archives** window. (The data for all robot controllers selected in the **Cell view** window can be archived.)

Device properties window:
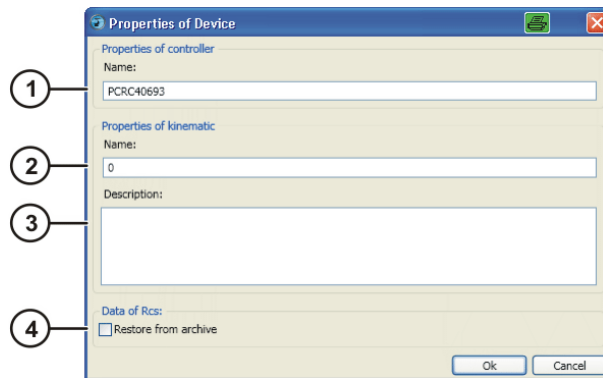
Fig. 14-15: "Device properties" window

Item	Description
1	The name of the robot controller can be changed here.
2	The name of the robot can be changed here.
3	A description can be entered here for information purposes. The description is displayed in the following places in the **Project deployment** window: ■ In the **Information** area ■ During activation in the lower window with the progress bar
4	**Activated**: If **OK** is pressed, the RDC data are transferred to the RDC memory from D:\BackupAll.zip.

Generate archives window:
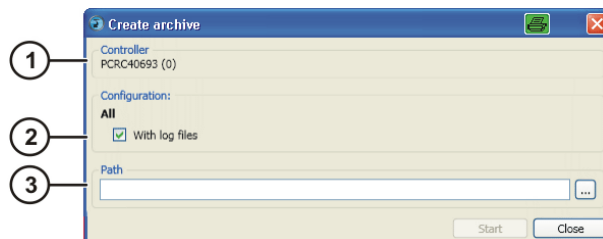
Fig. 14-16: "Generate archives" window

Item	Description
1	The name of the robot controller is displayed here. If the window was opened via the **Archive all** button, the window displays all the robot controllers that are selected in the **Cell view** window.
2	**Activated**: The log data are also archived. **Deactivated**: The log data are not archived.
3	A target directory for the archive can be selected here. A ZIP file is generated as an archive for each robot controller. The name of ZIP file always contains the name of the robot and the robot controller.

15 KUKA Service

15.1 Requesting support

Introduction   This documentation provides information on operation and operator control, and provides assistance with troubleshooting. For further assistance, please contact your local KUKA subsidiary.

Information   **The following information is required for processing a support request:**

■ Description of the problem, including information about the duration and frequency of the fault

■ As comprehensive information as possible about the hardware and software components of the overall system

The following list gives an indication of the information which is relevant in many cases:

  ■ Model and serial number of the kinematic system, e.g. the manipulator

  ■ Model and serial number of the controller

  ■ Model and serial number of the energy supply system

  ■ Designation and version of the system software

  ■ Designations and versions of other software components or modifications

  ■ Diagnostic package **KrcDiag**:

    Additionally for KUKA Sunrise: Existing projects including applications

    For versions of KUKA System Software older than V8: Archive of the software (**KrcDiag** is not yet available here.)

  ■ Application used

  ■ External axes used

15.2 KUKA Customer Support

Availability   KUKA Customer Support is available in many countries. Please do not hesitate to contact us if you have any questions.

Argentina   Ruben Costantini S.A. (Agency)
Luis Angel Huergo 13 20
Parque Industrial
2400 San Francisco (CBA)
Argentina
Tel. +54 3564 421033
Fax +54 3564 428877
ventas@costantini-sa.com

Australia   KUKA Robotics Australia Pty Ltd
45 Fennell Street
Port Melbourne VIC 3207
Australia
Tel. +61 3 9939 9656
info@kuka-robotics.com.au
www.kuka-robotics.com.au

Belgium	KUKA Automatisering + Robots N.V.
	Centrum Zuid 1031
	3530 Houthalen
	Belgium
	Tel. +32 11 516160
	Fax +32 11 526794
	info@kuka.be
	www.kuka.be
Brazil	KUKA Roboter do Brasil Ltda.
	Travessa Claudio Armando, nº 171
	Bloco 5 - Galpões 51/52
	Bairro Assunção
	CEP 09861-7630 São Bernardo do Campo - SP
	Brazil
	Tel. +55 11 4942-8299
	Fax +55 11 2201-7883
	info@kuka-roboter.com.br
	www.kuka-roboter.com.br
Chile	Robotec S.A. (Agency)
	Santiago de Chile
	Chile
	Tel. +56 2 331-5951
	Fax +56 2 331-5952
	robotec@robotec.cl
	www.robotec.cl
China	KUKA Robotics China Co., Ltd.
	No. 889 Kungang Road
	Xiaokunshan Town
	Songjiang District
	201614 Shanghai
	P. R. China
	Tel. +86 21 5707 2688
	Fax +86 21 5707 2603
	info@kuka-robotics.cn
	www.kuka-robotics.com
Germany	KUKA Roboter GmbH
	Zugspitzstr. 140
	86165 Augsburg
	Germany
	Tel. +49 821 797-4000
	Fax +49 821 797-1616
	info@kuka-roboter.de
	www.kuka-roboter.de

France	KUKA Automatisme + Robotique SAS
	Techvallée
	6, Avenue du Parc
	91140 Villebon S/Yvette
	France
	Tel. +33 1 6931660-0
	Fax +33 1 6931660-1
	commercial@kuka.fr
	www.kuka.fr
India	KUKA Robotics India Pvt. Ltd.
	Office Number-7, German Centre,
	Level 12, Building No. - 9B
	DLF Cyber City Phase III
	122 002 Gurgaon
	Haryana
	India
	Tel. +91 124 4635774
	Fax +91 124 4635773
	info@kuka.in
	www.kuka.in
Italy	KUKA Roboter Italia S.p.A.
	Via Pavia 9/a - int.6
	10098 Rivoli (TO)
	Italy
	Tel. +39 011 959-5013
	Fax +39 011 959-5141
	kuka@kuka.it
	www.kuka.it
Japan	KUKA Robotics Japan K.K.
	YBP Technical Center
	134 Godo-cho, Hodogaya-ku
	Yokohama, Kanagawa
	240 0005
	Japan
	Tel. +81 45 744 7691
	Fax +81 45 744 7696
	info@kuka.co.jp
Canada	KUKA Robotics Canada Ltd.
	6710 Maritz Drive - Unit 4
	Mississauga
	L5W 0A1
	Ontario
	Canada
	Tel. +1 905 670-8600
	Fax +1 905 670-8604
	info@kukarobotics.com
	www.kuka-robotics.com/canada

Korea	KUKA Robotics Korea Co. Ltd.
	RIT Center 306, Gyeonggi Technopark
	1271-11 Sa 3-dong, Sangnok-gu
	Ansan City, Gyeonggi Do
	426-901
	Korea
	Tel. +82 31 501-1451
	Fax +82 31 501-1461
	info@kukakorea.com
Malaysia	KUKA Robot Automation (M) Sdn Bhd
	South East Asia Regional Office
	No. 7, Jalan TPP 6/6
	Taman Perindustrian Puchong
	47100 Puchong
	Selangor
	Malaysia
	Tel. +60 (03) 8063-1792
	Fax +60 (03) 8060-7386
	info@kuka.com.my
Mexico	KUKA de México S. de R.L. de C.V.
	Progreso #8
	Col. Centro Industrial Puente de Vigas
	Tlalnepantla de Baz
	54020 Estado de México
	Mexico
	Tel. +52 55 5203-8407
	Fax +52 55 5203-8148
	info@kuka.com.mx
	www.kuka-robotics.com/mexico
Norway	KUKA Sveiseanlegg + Roboter
	Sentrumsvegen 5
	2867 Hov
	Norway
	Tel. +47 61 18 91 30
	Fax +47 61 18 62 00
	info@kuka.no
Austria	KUKA Roboter CEE GmbH
	Gruberstraße 2-4
	4020 Linz
	Austria
	Tel. +43 7 32 78 47 52
	Fax +43 7 32 79 38 80
	office@kuka-roboter.at
	www.kuka.at

Poland	KUKA Roboter Austria GmbH
	Spółka z ograniczoną odpowiedzialnością
	Oddział w Polsce
	Ul. Porcelanowa 10
	40-246 Katowice
	Poland
	Tel. +48 327 30 32 13 or -14
	Fax +48 327 30 32 26
	ServicePL@kuka-roboter.de

Portugal	KUKA Sistemas de Automatización S.A.
	Rua do Alto da Guerra n° 50
	Armazém 04
	2910 011 Setúbal
	Portugal
	Tel. +351 265 729780
	Fax +351 265 729782
	kuka@mail.telepac.pt

Russia	KUKA Robotics RUS
	Werbnaja ul. 8A
	107143 Moskau
	Russia
	Tel. +7 495 781-31-20
	Fax +7 495 781-31-19
	info@kuka-robotics.ru
	www.kuka-robotics.ru

Sweden	KUKA Svetsanläggningar + Robotar AB
	A. Odhners gata 15
	421 30 Västra Frölunda
	Sweden
	Tel. +46 31 7266-200
	Fax +46 31 7266-201
	info@kuka.se

Switzerland	KUKA Roboter Schweiz AG
	Industriestr. 9
	5432 Neuenhof
	Switzerland
	Tel. +41 44 74490-90
	Fax +41 44 74490-91
	info@kuka-roboter.ch
	www.kuka-roboter.ch

Spain	KUKA Robots IBÉRICA, S.A.
	Pol. Industrial
	Torrent de la Pastera
	Carrer del Bages s/n
	08800 Vilanova i la Geltrú (Barcelona)
	Spain
	Tel. +34 93 8142-353
	Fax +34 93 8142-950
	Comercial@kuka-e.com
	www.kuka-e.com
South Africa	Jendamark Automation LTD (Agency)
	76a York Road
	North End
	6000 Port Elizabeth
	South Africa
	Tel. +27 41 391 4700
	Fax +27 41 373 3869
	www.jendamark.co.za
Taiwan	KUKA Robot Automation Taiwan Co., Ltd.
	No. 249 Pujong Road
	Jungli City, Taoyuan County 320
	Taiwan, R. O. C.
	Tel. +886 3 4331988
	Fax +886 3 4331948
	info@kuka.com.tw
	www.kuka.com.tw
Thailand	KUKA Robot Automation (M)SdnBhd
	Thailand Office
	c/o Maccall System Co. Ltd.
	49/9-10 Soi Kingkaew 30 Kingkaew Road
	Tt. Rachatheva, A. Bangpli
	Samutprakarn
	10540 Thailand
	Tel. +66 2 7502737
	Fax +66 2 6612355
	atika@ji-net.com
	www.kuka-roboter.de
Czech Republic	KUKA Roboter Austria GmbH
	Organisation Tschechien und Slowakei
	Sezemická 2757/2
	193 00 Praha
	Horní Počernice
	Czech Republic
	Tel. +420 22 62 12 27 2
	Fax +420 22 62 12 27 0
	support@kuka.cz

Hungary   KUKA Robotics Hungaria Kft.

Fö út 140

2335 Taksony

Hungary

Tel. +36 24 501609

Fax +36 24 477031

info@kuka-robotics.hu

USA   KUKA Robotics Corporation

51870 Shelby Parkway

Shelby Township

48315-1787

Michigan

USA

Tel. +1 866 873-5852

Fax +1 866 329-5852

info@kukarobotics.com

www.kukarobotics.com

UK   KUKA Robotics UK Ltd

Great Western Street

Wednesbury West Midlands

WS10 7LL

UK

Tel. +44 121 505 9970

Fax +44 121 505 6589

service@kuka-robotics.co.uk

www.kuka-robotics.co.uk

Index