Identification number: CS382

KUKA.Sim 4.3

Specifying the supply voltage and controller version

Procedure

1. FILE area > **Options** > **General** > **Machine data service**
   - **Supply voltage**: Select the required voltage.
   - **Controller version**: Select the required controller version.

   If a robot that is not compatible with this version is subsequently used in the 3D scene, KUKA.Sim uses the closest of the compatible versions for this robot. A message in the output window indicates this.

2. Confirm with **OK**.

Exporting the WorkVisual project

Description

KUKA.Sim automatically creates a WorkVisual project in the background for each Sim project. This can be exported as a WVS file and opened in WorkVisual.

Procedure

1. START area > **Export** group > **Export project**
2. Set parameters for export.
3. Confirm with **Export** . The WVS file is saved to the selected path.

Parameters

Overwrite

- Check box active: If a WVS file of the same name already exists in the selected path, it will be overwritten without prompting.
- Check box not active: If a WVS file of the same name already exists in the selected path, the export is not possible.

Include scene

- Check box active:

  The 3D scene is also exported. This means that it is available again in KUKA.Sim if the WorkVisual project is subsequently reopened in KUKA.Sim.
  The 3D scene cannot be displayed in WorkVisual itself and is not relevant.
- Check box not active: The 3D scene is not exported.

Import field bus configuration

Description

PROFINET and EtherCAT devices and their I/O mappings can be imported into KUKA.Sim. For this, a subproject containing these devices must be exported in WorkVisual.

Precondition

- Subproject with the field bus configuration, exported from WorkVisual
- Robot present in 3D scene
- START area

Procedure

1. Right-click on the **Field bus**node in the **Devices** window and select **Import field bus configuration**.

2. Confirm the message indicating that the current field bus configuration is being deleted with **OK**. A window opens.
3. Navigate to the path where the subproject is located and select it.
4. Click on **Open**. The field bus configuration is imported; the imported devices are inserted under the node **Field bus**.

KUKA option packages

Installing or uninstalling a KUKA option package

Description

KUKA option packages are supplied as KOP files. In order to be able to use them, they must first be installed and then assigned to the robot controller.

Installation

1. FILE area > **Options** > **Add-ons**
2. Click on **Manage** under **KUKA Option packages**. The **Option package management** window opens.
3. Click on the  button. The **Select package to be installed** window opens. Navigate to the path where the option package is located and select it. Click on **Open**.

   Alternatively: Drag the desired option package or multiple option packages from Windows Explorer to the “Option package management” window.
4. The **Installing {0} option package(s)** window is opened and the package is installed. Once the operation has been completed, the package is displayed in the **Option package management** window.
5. Only if the message **The application must be restarted to complete the installation.** is displayed: click on the **Restart** button. KUKA.Sim restarts.
6. Only if the message stated in the previous step is NOT displayed: close the **Option package management** window.

Uninstallation

1. FILE area > **Options** > **Add-ons**
2. Click on **Manage** under **KUKA Option packages**. The **Option package management** window opens.
3. Click on the  button. The option package is uninstalled.
4. Only if the message **The application must be restarted to complete the installation.** is displayed: click on the **Restart** button. KUKA.Sim restarts.
5. Only if the message stated in the previous step is NOT displayed: close the **Option package management** window.

Assigning a KUKA option package

Description

In order to be able to use the functions of an option package, the option package must be assigned to the Sim project.

There are several ways of assigning an option package to the Sim project in KUKA.Sim:

- Using a component from the **eCatalog** window
- Via the **Component properties** window of the robot

A component of the option package can be assigned in 2 ways:

- By attaching the component to the robot
- By connecting the interfaces of component and robot

Not all option packages bring a component for the 3D scene. In these cases, assignment is only possible via the **Component properties** window.

Precondition

- The option package is installed.
- Robot present in 3D scene
- START area

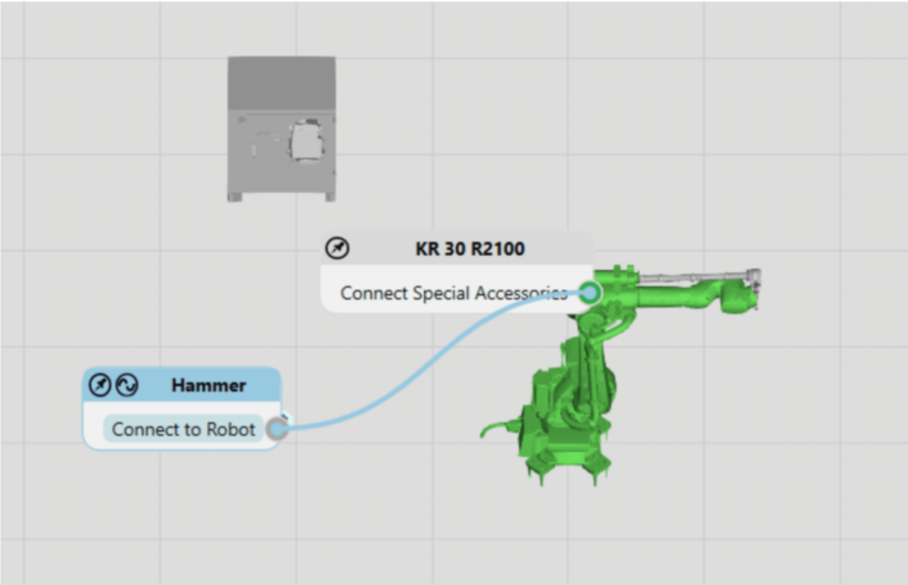
Procedure for attaching the component

1. In the **eCatalog** window, expand the collection of **option packages** and select the folder of the option package. The corresponding components are displayed in the right-hand area of the window.
2. Drag the component into the 3D scene and select it.

3. Drag the component to the robot to attach it. A window opens. The files being added or modified are indicated in the window.
4. Click on **OK**. The window closes and the option package is assigned to the project.

Procedure for connecting the component

1. In the **eCatalog** window, expand the collection of **option packages** and select the folder of the option package. The corresponding components are displayed in the right-hand area of the window.
2. Drag the component into the 3D scene and select it.
3. Select ribbon > **Connect** group > **Interfaces**.
4. Drag the **Connect Special Accessories** node of the robot to the **Connect to Robot** node of the component of the option package to connect it. A window opens. The files being added or modified are indicated in the window.



Assignment of the option package via interfaces

5. Click on **OK**. The window closes and the option package is assigned to the project.

Procedure using “Component properties” window

1. Select the robot controller or robot in the 3D scene.
2. In the right-hand area of the **Component properties** window, click on **Option packages configuration**. The available option packages are displayed.
3. Click on + next to the option package. A window opens. The files being added or modified are indicated in the window.
4. Click on **OK**. The window closes and the option package is assigned to the project.

Removing a KUKA option package

Description

This procedure is used to remove an option package from the Sim project.

Precondition

- The option package has been installed and assigned.
- Robot present in 3D scene
- START area

Procedure

> When the option package is removed, the data belonging to the option package are deleted.

1. Select the robot controller or robot in the 3D scene.
2. On the right-hand side of the **Component properties** window, click on **Option packages configuration**. The assigned option packages are displayed.
3. Click on 🗑 next to the option package. A window opens. The files being deleted are indicated in the window.
4. Click on **OK**. The window closes and the option package is removed from the project.

“**Devices**” window

Description

The **Devices** window is displayed in the START area by default. It can also be included in the view of the other areas.

The **Devices** window enables the following:

- Activating the controller

  (>>> Activating the robot controller / displaying devices)

- Overview of the cell layout
- Hiding and showing components in the 3D scene
- Deleting components in the window and in the 3D scene
- Locking components (protection against unintentional modification or deletion) in the window and in the 3D scene
- Focus on the component in the 3D scene by double-clicking on the component in the device tree
- Calling the safety configuration
- Calling actions for the cell components via context menu

  Which if any actions are available depends on the individual component and the specific cell layout.

- Showing hidden components

The components can be displayed in one of two ways:

- By hierarchy

  The representation is in the form of a structure tree. The tree contains:

  - Cell components: controllers, robots, field buses including their subordinate devices, external axes

    Components that are generally never visible in the 3D scene, e.g. internal bus components such as the CIB, are also shown.

  - Safety configuration
  - Further components of the 3D scene, e.g. energy supply systems, pallets, people

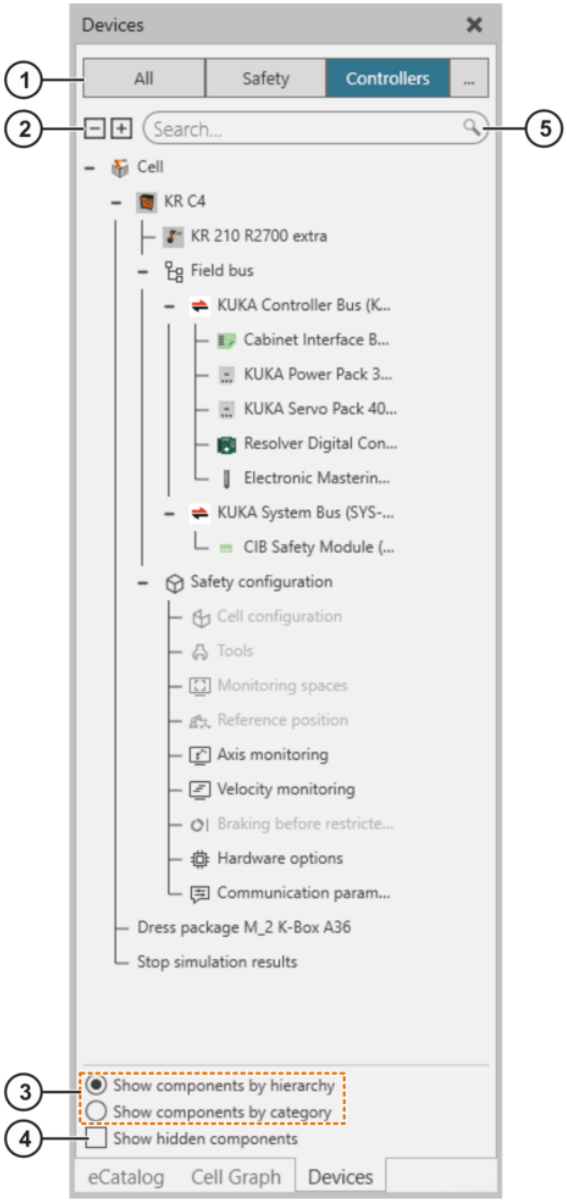
- By category

  The display is in the form of a list; the components are assigned to the individual categories. The list contains the following categories, for example:

  - Controllers
  - Robots
  - Field bus
  - Safety configuration

A component that the user selects in the tree is automatically also selected in the 3D scene (if available there) and vice versa.

Overview



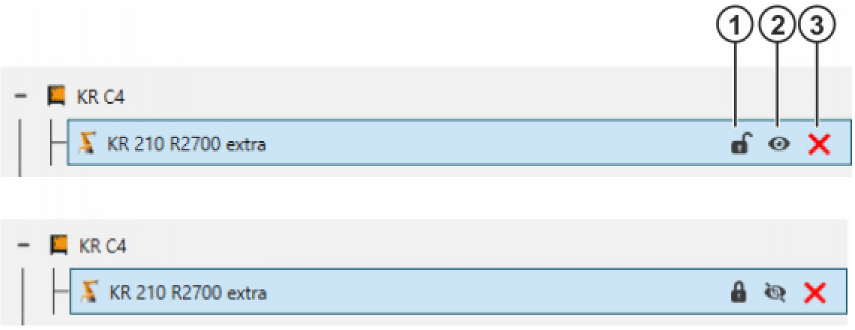
"**Devices**" window

Item	Description
1	Display filter Only the components selected by a filter are displayed in the tree. The active filter is highlighted in blue. Clicking on "**...**" (far right) displays further filters. Which filters are displayed depends on the components contained in the tree. For example, there can be filters for components from an option package.
2	Clicking on "**+**" displays all components. Clicking on "**-**" hides all components. In the display by hierarchy, only the **Cell** node is displayed. In the display by category, only the categories remain displayed.
3	Display of components by hierarchy or by category
4	Display option for hidden components
5	Full text search within the filtered scope

Icons: Lock, Hide, Delete

The icons become visible in the **Devices** window when you hover over the respective level with the mouse.

They are only available, however, for components that can generally be visible in the 3D scene. This does NOT include internal bus components, such as the CIB, for example.



Icons: Lock, Hide, Delete

Item	Description
1	Upper screenshot: not locked Clicking on the icon locks the component in the **Devices** window and in the 3D scene. A locked component cannot be moved, edited or deleted.
	Lower screenshot: locked Clicking on the icon unlocks the component.
2	Upper screenshot: displayed Clicking on the icon hides the component in the 3D scene.
	Lower screenshot: hidden Clicking on the icon displays the component again.
3	Clicking on the “X” deletes the component from the tree and from the 3D scene.

Activating the robot controller / displaying devices

Description

If a robot has been newly inserted into the 3D scene, it will also be displayed in the **Devices** window, including the robot controller. To make all associated nodes (e.g. field bus devices) visible there as well, the robot controller must be activated and the project reopened.

Procedure

1. **Devices** window > right-click on the robot controller > **Activate controller**
2. Save.
3. Close the project and open it again:
    1. FILE area > **New**. The project closes and an empty project opens.
    2. FILE area > reopen the previous project.

       All devices are now displayed in the **Devices** window.

If the project is closed and opened again, the robot controller remains active and the devices remain visible. It is not necessary to activate the robot controller again.

“**I/O mapping editor**” window

Call

1. START, MODELING or PROGRAM area
2. Ribbon > **Connect** group > **I/O mapping editor**

Alternatively:

1. START area
2. **Devices** window > right-click on the robot controller > **Open I/O mapping**

Description

The **I/O mapping editor** window enables the following:

- Mapping signals

  (>>> View: mapped signals)
- Mapping simulated signals

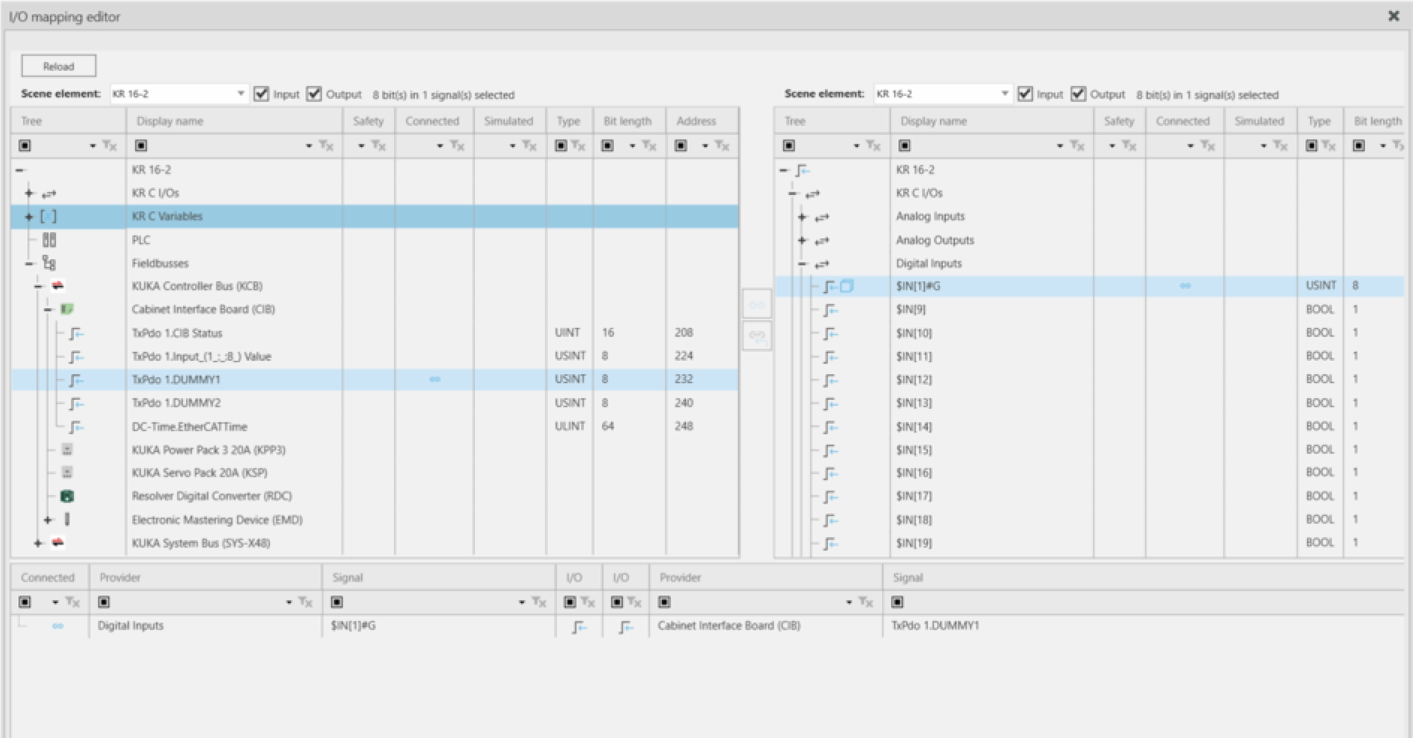
  (>>> View: mapped simulated signals)

Numerous support functions are available for this purpose, e.g.:

- Searching with search criteria in each column
- Filter for inputs or outputs

- Editing signals: group, split, swap
- Renaming I/Os of field bus devices
- Differentiated display of signal types and states via icons or highlighting

Overview



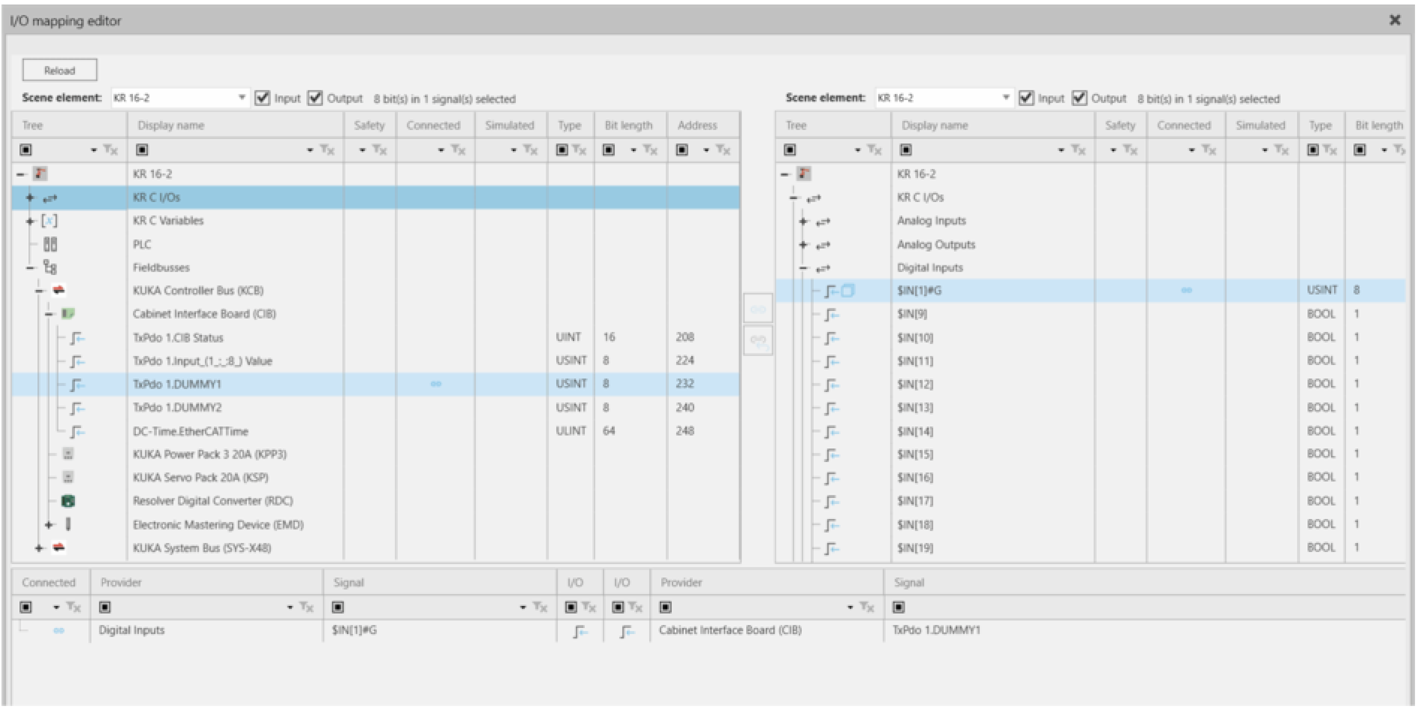
Overview

Designation	Description
Reload	Refreshes the **I/O mapping editor** window according to the current 3D scene. After modifications to the 3D scene, a message flashes next to **Reload** as a reminder to refresh the window.
Left and right areas:	
Scene element:	Select which element of the 3D scene is to be displayed.
Input	Check box active: Display is filtered for inputs
Output	Check box active: Display is filtered for outputs
Columns:	
Tree	Representation of the selected element and the associated devices in the form of a tree structure To display the I/Os of a device, click on the device and open further levels if necessary.
Display name	Name of the signal I/Os of field bus devices can be renamed.
Safety	: Safe signals are indicated by this icon.
Connected	: Mapped signals (but not mapped simulated signals) are indicated by the chain icon. Signals with multiple mapping are indicated by a longer chain icon.
Simulated	: Simulated signals are indicated by the yellow lightning icon.
Type, **Bit length**, **Address**	The associated information is displayed.
Icons in the middle:	
Mapping	Select the signals to be mapped on the left and right. : maps the selected signals. **Note**: The icon is only active if the selected signals can be mapped. It is not active, for example, if signals with different bit lengths have been selected.
Unmap	: disconnects the mapping selected in the lower area
Lower area: Display of the mapped signals	

View: mapped signals

The mapping corresponds to the mapping familiar from WorkVisual: Signals that are located beneath the same robot controller can be mapped.

These mappings are then also present in the integrated WorkVisual project.



View: mapped signals

The same robot is selected on the left and right under **Scene element:**.

- In both areas, all signals belonging to the robot are shown in the **Tree** column.

In both areas, the mapped signals are indicated in the **Connected** column by the blue chain icon.

The lower area indicates which signal is mapped to which.

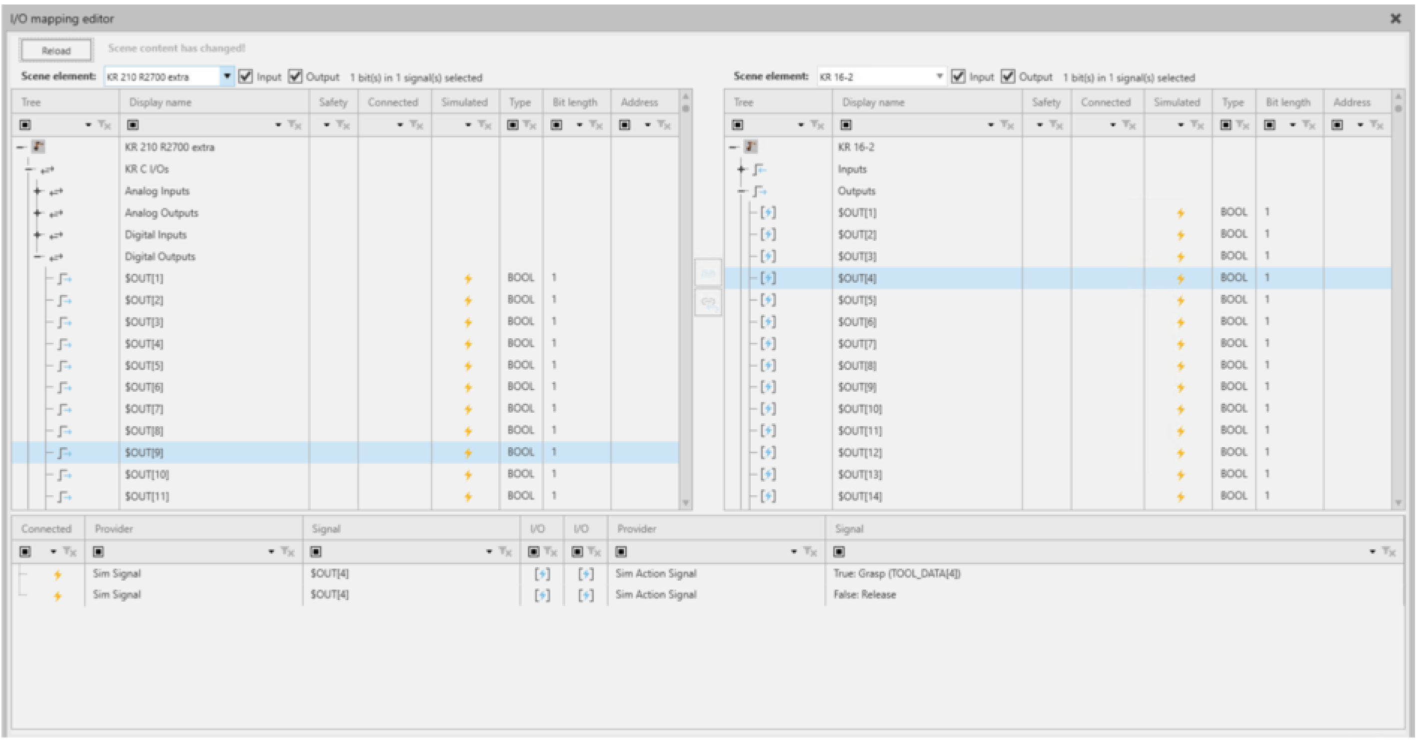
The row with the mapping is selected in the lower area:

- The **Mapping** icon is grayed out, as mapping has already been carried out.
- The **Unmap** icon is active.

View: mapped simulated signals

Signals can be mapped between various elements of the 3D scene.

These mappings are only present in KUKA.Sim.



View: mapped simulated signals

Various elements on the left and right are selected under **Scene element:**.

- All signals belonging to the robot are displayed on the left in the **Tree** column.
- The simulated signals are displayed on the right.

Simulated signals are indicated by a yellow lightning icon in the **Simulated** column:

- Left: signals that are mapped
- Right: all signals

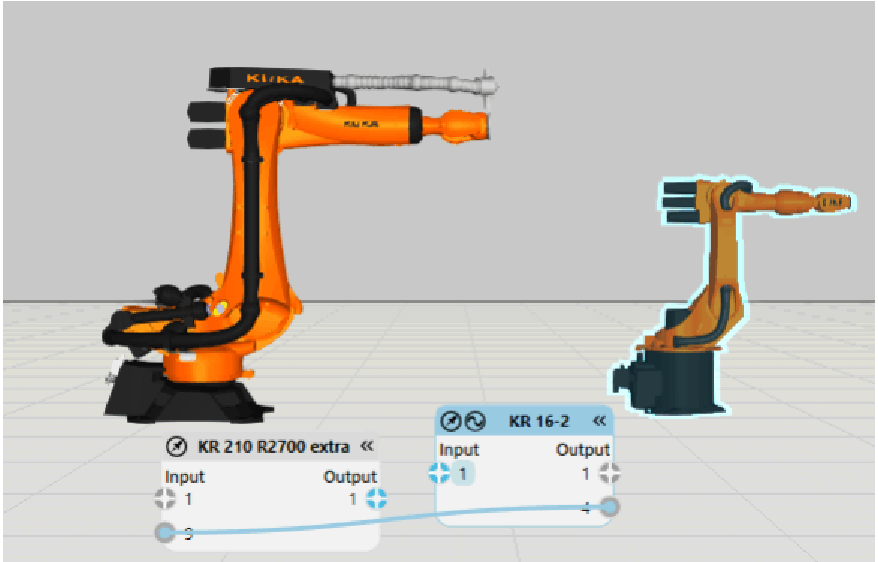
The lower area indicates which signal is mapped to which.

The row with the mapping is selected in the lower area:

- The **Mapping** icon is grayed out, as mapping has already been carried out.
- The **Unmap** icon is active.

“I/O mapping editor” window / 3D scene

The display of mapped simulated signals is synchronized between the **I/O mapping editor** window and the 3D scene.



Mapped simulated signals in the 3D scene

Displaying the mappings in the 3D scene:

1. START, MODELING or PROGRAM area
2. Ribbon > **Connect** group > **Signals**

Searching for a signal / searching in columns

Description

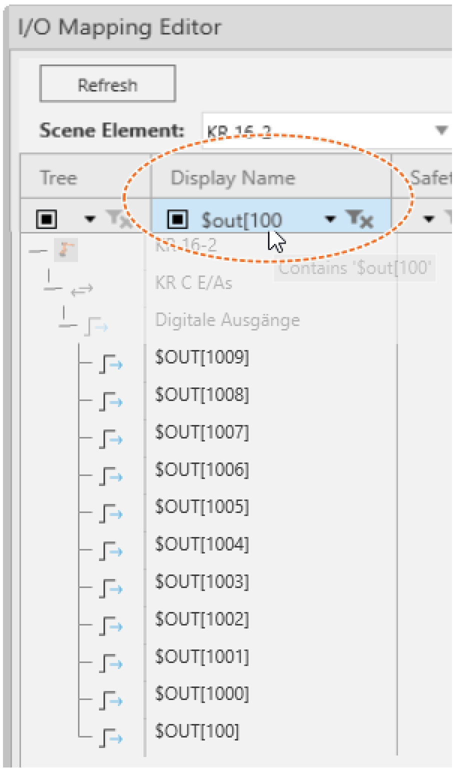
The search is described here for signal names. The same principle can be used to search in all columns.

Precondition

- “**I/O mapping editor**” window

Procedure

1. Click in the empty area below **Display name**. The area becomes editable.
2. Enter the search term. Only matching signals are displayed.
3. The search criterion can be changed by clicking on the box to the left of the search box. The default criterion is **Contains**.



Example: searching for signals containing “$out[100”

Renaming I/Os

Description

I/Os of field bus devices can be renamed.

Precondition

- “**I/O mapping editor**” window

Procedure

1. Display the **Properties** window if it is not already displayed.
2. If necessary, move the **I/O mapping editor** window to the side so that both windows can be viewed at the same time.
3. In the **I/O mapping editor** window, select the input or output that is to be renamed.
4. In the **Properties** window, change the name and confirm with ENTER.

   The change is immediately displayed in the **I/O mapping editor** window.

Group signals

Description

Digital inputs or outputs of the robot controller can be grouped together to form a signal of data type SINT, USINT or BYTE. 8 signals or multiples of 8 can be grouped.

Grouped signals can be recognized by the name extension **#G**.

Precondition

- **I/O mapping editor** window
- The I/Os that are to be grouped are not mapped.

Procedure

Group:

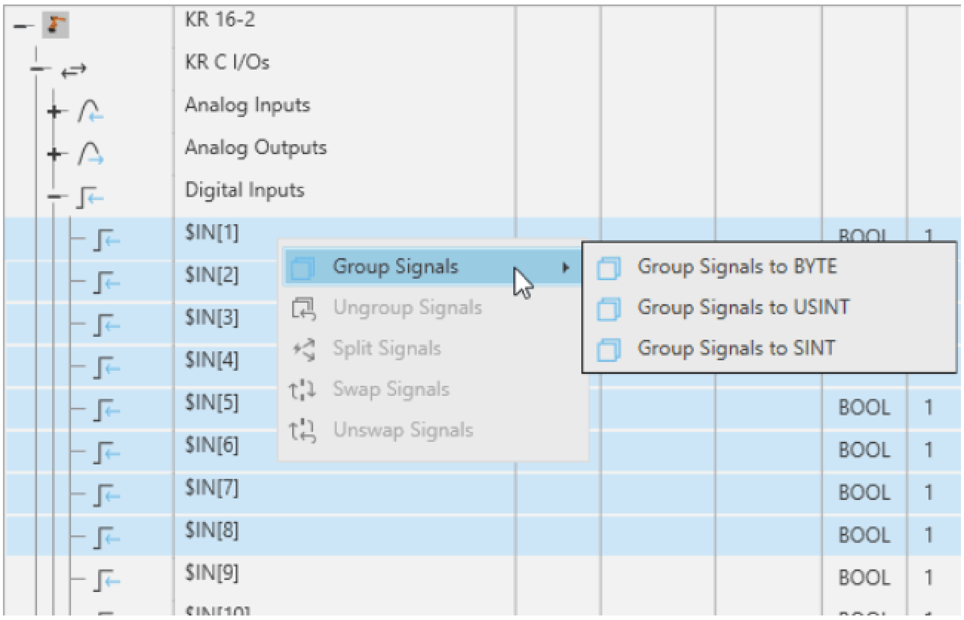
- Select successive signals and right-click on them. Select > **Group signals** > data type.

The signals are grouped to a signal of the selected data type.

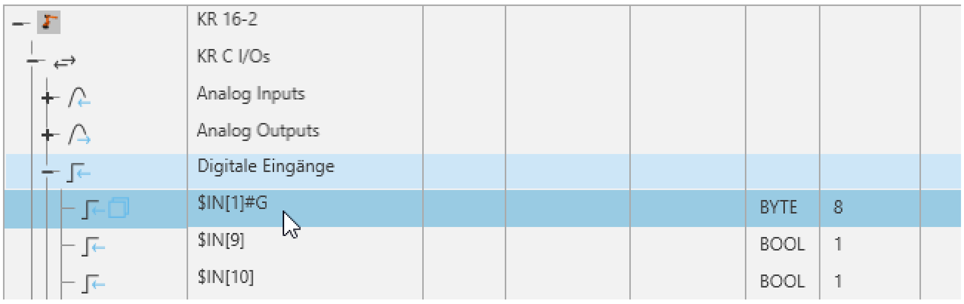
Undo:

- Right-click on a signal with **#G**. > **Ungroup signals**

Example



Group signals



A grouped signal

Splitting signals

Description

Signals with a bit length of 8 or more can be split.

Precondition

- "**I/O mapping editor**" window

Procedure

Splitting single signal:

- Right-click on the signal > **Split signal** > select desired submenu item

Which submenu items (=splitting options) are available depends on the signal. Always possible: **Split signal to BOOL**

Split multiple signals at once:

1. Select multiple signals. They do not have to be successive signals.
2. Right-click > **Split signal** > select desired submenu item

Which submenu items are available depends on the smaller signal.

Configuring Fast Measurement inputs

Description

This procedure is used to connect the Fast Measurement inputs to the inputs of digital sensors.

The Fast Measurement inputs are activated as standard, so it is not generally necessary to activate them.

Precondition

- Robot and sensor present in 3D scene
- START area

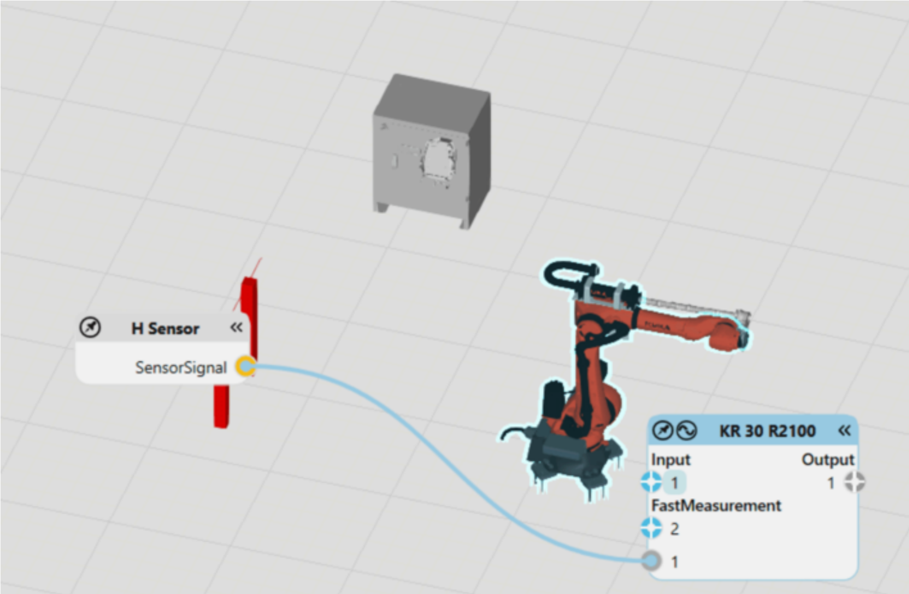
Preparation

If the Fast Measurement inputs happen not to be activated, they can be activated using this procedure:

1. Select the robot in the 3D scene.
2. Select the **Controller** tab in the **Component properties** window.
3. Set the check mark in the box **Fast Measurement input**.

Procedure

1. Select the robot in the 3D scene.
2. Select ribbon > **Connect** group > **Signals**.
3. Drag the **FastMeasurement** node of the robot to the **SensorSignal node** of the sensor to connect it.



Connecting Fast Measurement inputs

Safety configuration

Safety configuration in KUKA.Sim

A valid safety configuration is a precondition for the simulation with robots to be started. For certain configurations, e.g. multiple safety-oriented tools, a safety option must be assigned to the project and, if necessary, the required safety interfaces must be configured. If this is not the case, the simulation cannot be started, or can only be started after the **Safe monitoring** function has been deactivated. The safety configuration cannot be edited while a simulation is running.

The current safety configuration can be printed out or an existing safety configuration can be imported from a .scg or .xml file. To do so, right-click on **Safety configuration** in the **Devices** window and select the corresponding option. In order to be able to import an .xml file, the safety option SafeOperation or SafeRangeMonitoring must be assigned to the project.

Displaying information about the safety configuration

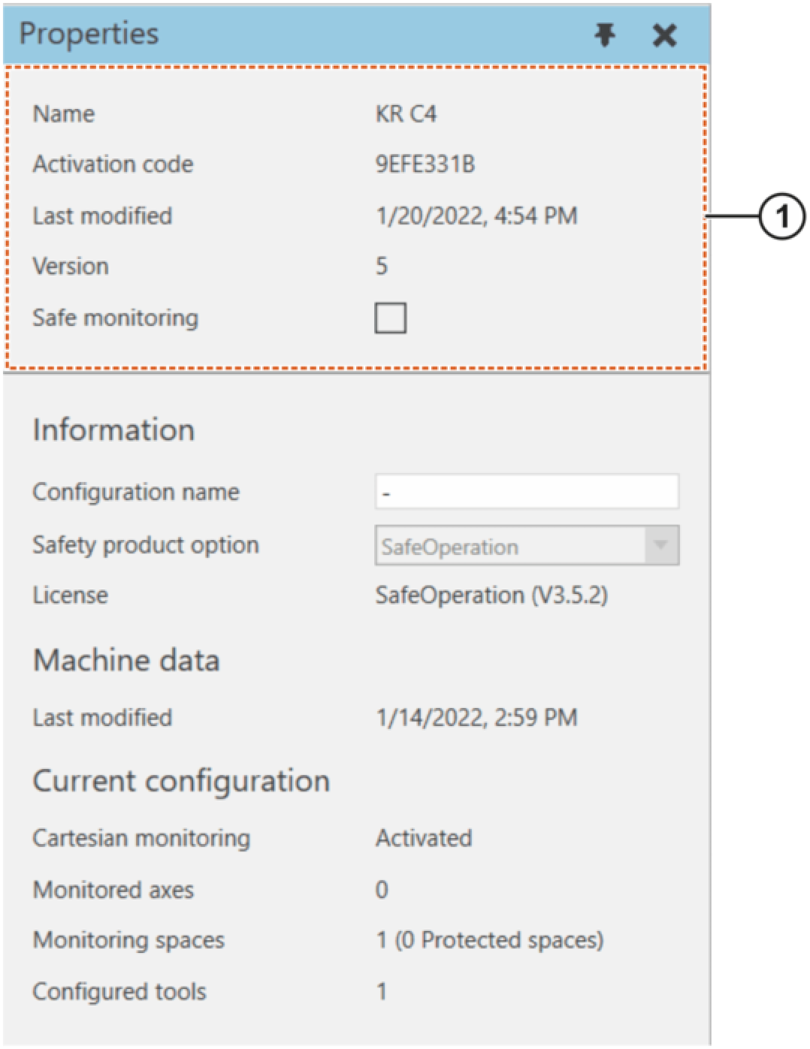
Precondition

- Robot present in 3D scene
- START area

Procedure

- In the **Devices** window, click on the **Safety configuration** node. The corresponding information is displayed in the **Properties** window.

Information about the safety configuration



Information about the safety configuration

Item	Description
1	This area is also visible for all subordinate nodes of the safety configuration. Safe monitoring can also be activated or deactivated accordingly for all subordinate nodes.

Boxes

Box	Description
Name	Name of the robot controller
Activation code	Activation code of the safety configuration If a change has been made that affects the activation code, a warning symbol is displayed next to the activation code.
Last modified	Last modification of the safety configuration (date and time last saved)
Version	Version of the safety configuration
Safe monitoring	• Check box active: Safe monitoring is activated. • Check box not active: Safe monitoring is not activated.
Information	
Configuration name	Name of the safety configuration
Safety product option	Assigned safety option If no safety option has been assigned to the project, test versions of the safety options are available for selection in this box. If a test version is selected, it is assigned to the project. The test version can be removed again by selecting the option **Standard**. • **Standard**: No safety option has been assigned to the project. • **SafeSingleBrake**: Test version of the safety option KUKA.SafeSingleBrake • **SafeRangeMonitoring**: Test version of the safety option KUKA.SafeRangeMonitoring • **SafeVelocityMonitoring**: Test version of the safety option KUKA.SafeVelocityMonitoring • **SafeOperation**: Test version of the safety option KUKA.SafeOperation If a regular version has been assigned, this box is grayed out. In order to be able to use a test version in this case, the regular version must be removed from the project.
License	License of the assigned safety option Whether a regular version or a test version has been assigned is also indicated here.
Machine data	
Last modified	Last modification of the safety-relevant machine data (date and time last saved)
Current configuration	
Cartesian monitoring	State of Cartesian monitoring (= velocity monitoring in T1) • **Activated** • **Deactivated**
Monitored axes	Number of velocity-monitored axes
Monitoring spaces	Number of monitoring spaces
Configured tools	Number of safety-oriented tools

Activate safe monitoring

Safe monitoring can only be activated in the following cases:

- The option package KUKA.SafeOperation or a different safety option has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or another safety option has been assigned to the project.

Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the window **Devices**, click on **Safety configuration** or a node underneath it. The corresponding setting is displayed in the **Properties** window.
2. Set the check mark in the check box **Safe monitoring**.

Configuring a cell area

Configuration is only possible in the following cases:

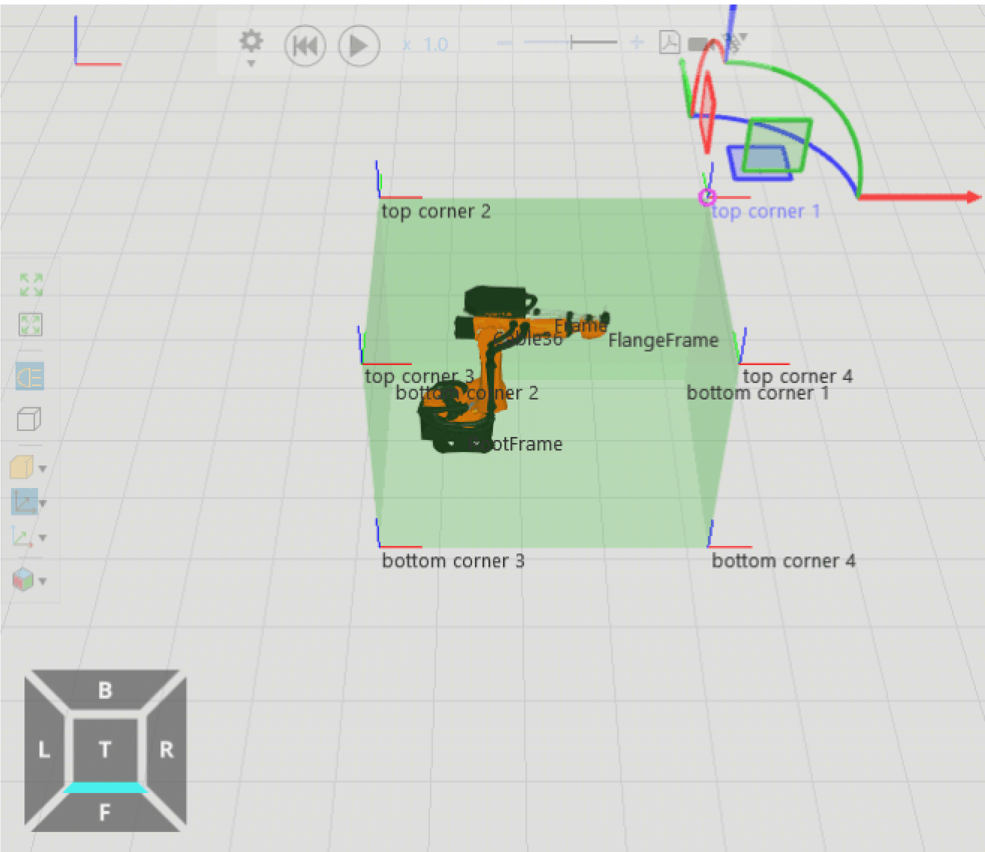
- The option package KUKA.SafeOperation has been installed and assigned to the project.

- A test version of KUKA.SafeOperation has been assigned to the project.

Description

The **Devices** window contains a cell for each robot.

- The cell cannot be deleted.
- In the 3D scene, the cell is displayed as standard when a safety option is installed. Otherwise, the cell is hidden as standard.



3D scene with configured cell displayed

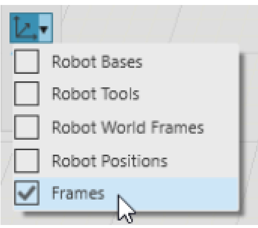
- Cell green: Outline of the cell OK
- Cell red: The defined corners form a convex space. The outline must be modified.
- Cell gray: Safe monitoring is not activated.

Precondition

- Robot present in 3D scene
- START area

Procedure

1. Activate “Frame” in the visualization settings in the 3D scene. This displays the components with labels, in particular the corners of the cell.

2. In the **Devices** window, click on **Cell configuration** under **Safety configuration**.

   The corresponding settings are displayed in the **Component properties** window. The cell is displayed in the 3D scene. As standard, the cell is very large and superposed on the 3D scene until it is configured smaller.

3. Configure the cell.

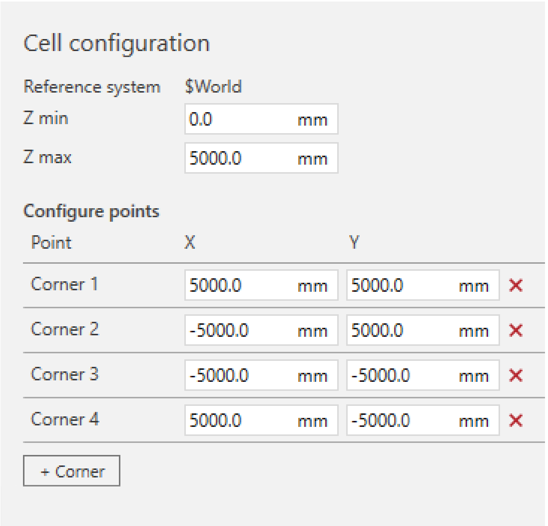
   - Enter the required values under **Cell configuration**. The changes are displayed in the 3D scene.
   - And/or: configure the cell in the 3D scene.

     The shift tool can be used, for example, to grab the corners with the mouse pointer and shift them.

     To use the shift tool, select the following in the ribbon: **Handling** group > **Shift**.

     To place the shift tool at a corner, click on **Corner {No.}** in the **Cell configuration** window.

Cell configuration



Cell configuration

Box	Description
Reference system	**Z min**: lower limit of the cell area, "Floor" "0" corresponds to the real cell floor, but the cell area can also start at a lower position (negative value). **Z max**: upper limit, "Ceiling"
Corner {No.}	Define X and Y values of the corner. Each corner automatically exists at the top and at the bottom, i.e. on the floor and ceiling of the cell. It thus actually forms an edge. **Note:** The number has no relation to the position of the corner in the 3D scene. For example, if you grab corner 5 and drag it between corners 3 and 4, it will remain corner 5. **X**: Deletes the corner.
Add corner	Adds a corner. Up to 16 corners are possible.

Configuring a safety-oriented tool

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation has been installed and assigned to the project.
- A test version of KUKA.SafeOperation has been assigned to the project.

Description

The **Devices** window contains up to 16 tools for each robot.

- Tool 1 is displayed as standard in the 3D scene when a safety option is installed. Otherwise, the tool is hidden as standard.
- Tools 2 to 16 are without spheres by default and therefore inactive.

  They are hidden in the **Devices** window and in the 3D scene.
- The tools can have the following colors:
  - Yellow: Tool is active
  - Light yellow: Tool is not active
  - Gray: Safe monitoring is not activated

Precondition

- Robot present in 3D scene
- START area

Procedure

Display overview of monitored tools:

- In the **Devices** window, click on the **Tools** node.

  The corresponding settings are displayed in the **Properties** window.

Configuring tool 1:

1. Expand the **Tools** node in the **Devices** window and click on **Tool 1**.

   The corresponding settings are displayed in the **Component properties** window.

2. Configure the tool.
   - Enter the required values under **Tool** or **Spheres**. The changes are displayed in the 3D scene.
   - Some parameters can also be configured in the 3D scene:

     The shift tool can be used, for example, to grab the sphere with the mouse pointer and shift it.

     To use the shift tool, select the following in the ribbon: **Handling** group > **Shift**.

     To place the shift tool on a sphere, click on the sphere in the 3D scene.

Configuring other tools:

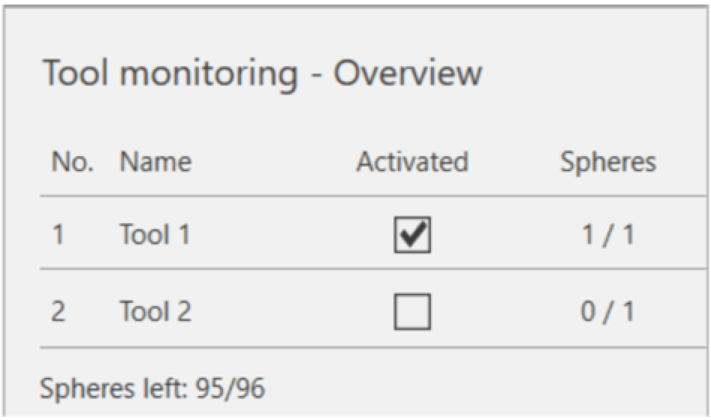
1. Define which tools are to be displayed in the **Devices** window:

   Right-click on **Tools** > **Show active tools only** (default) or **Show all tools**
2. Add another tool:
   - Right-click on **Tools** > **Add new tool**:

     The next tool in numerical order is displayed. The corresponding settings can be made in the **Component properties** window.
   - Or, if all the tools are visible: Double-click on any tool. The settings for this tool are displayed in the **Component properties** window.

   Once a tool has been set to active, it has sphere 1 as standard.
3. Configure the tool in the manner familiar from tool 1.

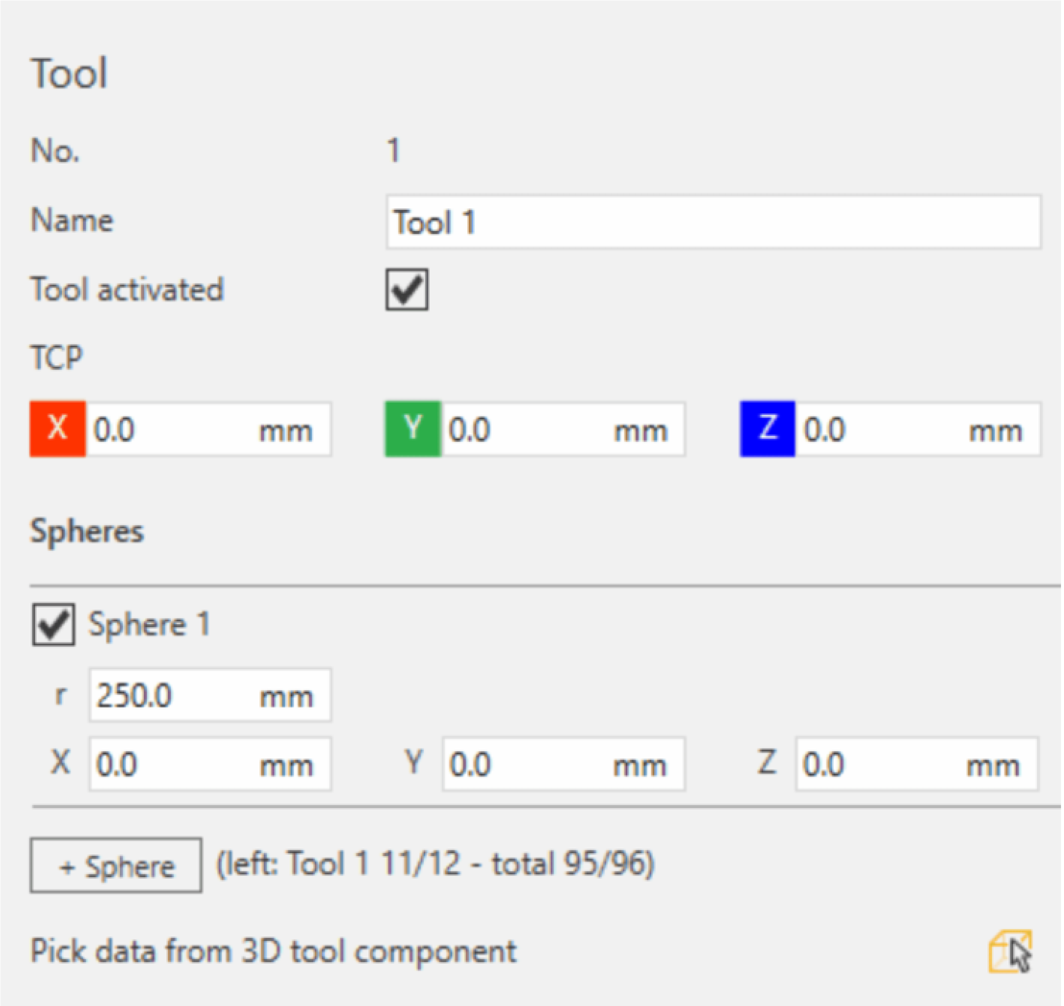
Tool monitoring - Overview



Tool monitoring - Overview

Box	Description
No.	Number of the tool
Name	Name of the tool
Activated	• Check box active: Tool is active • Check box not active: Tool is not active
Spheres	Number of spheres for the tool **Note**: Up to 12 spheres can be created for each tool. A total of up to 96 spheres can be created for all tools.

“Tool” and “Spheres”



"Tool" and "Spheres"

Box	Description
No.	Number of the tool
Name	Name of the tool, editable
Tool activated	Used to switch off a tool briefly during start-up, e.g. to take a closer look at another tool. **Note**: The setting has no influence on whether a tool is in the "active" or "inactive" list.
TCP	Position of the TCP, relative to the flange
Sphere {No.}	Sphere 1 on tool 1 cannot be deleted. All other spheres have an **X** here and can be deleted. If the last sphere is deleted, the tool becomes inactive. • Check box active: Sphere is active • Check box not active: Sphere is inactive **r**: Radius X, Y and Z values, relative to the TCP
+ Sphere	Adds a sphere. Up to 12 spheres can be created for each tool. A total of up to 96 spheres can be created for all tools.
Pick data from 3D tool component	After clicking on the button, a previously configured tool can be accepted as a safety-oriented tool in the 3D scene. The tool is framed in red when selected. (>>> Creating 3D objects as a template for the safety configuration)

Configuring a monitoring space

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been assigned to the project.

Description

The **Devices** window contains up to 16 monitoring spaces for each robot.

- The spaces are inactive by default.

  They are hidden in the **Devices** window and in the 3D scene.
- Active spaces are displayed in the 3D scene if the size has been configured.
- It is possible to configure whether the space is to be a workspace or a protected space.

Precondition

- Robot present in 3D scene
- START area

Procedure

Displaying the overview of the monitoring spaces:

- In the **Devices** window, click on the **Monitoring spaces** node.

  The corresponding settings are displayed in the **Properties** window.

Configuring the monitoring space:

1. Define which spaces are to be displayed in the **Devices** window:

   Right-click on **Monitoring spaces** > **Show active spaces only** (default) or **Show all spaces**
2. Configure space:
   - Right-click on **Monitoring spaces** > **Add new space**

     The space is added to the device tree and activated. Click on the space.
   - Or, if all spaces are visible: Double-click on any space.
3. The corresponding settings are displayed in the **Properties** window. Carry out these settings as required.

   The parameters displayed depend on whether the type is defined as “Cartesian space” or “Axis space”.

   Cartesian spaces are displayed in the 3D scene.

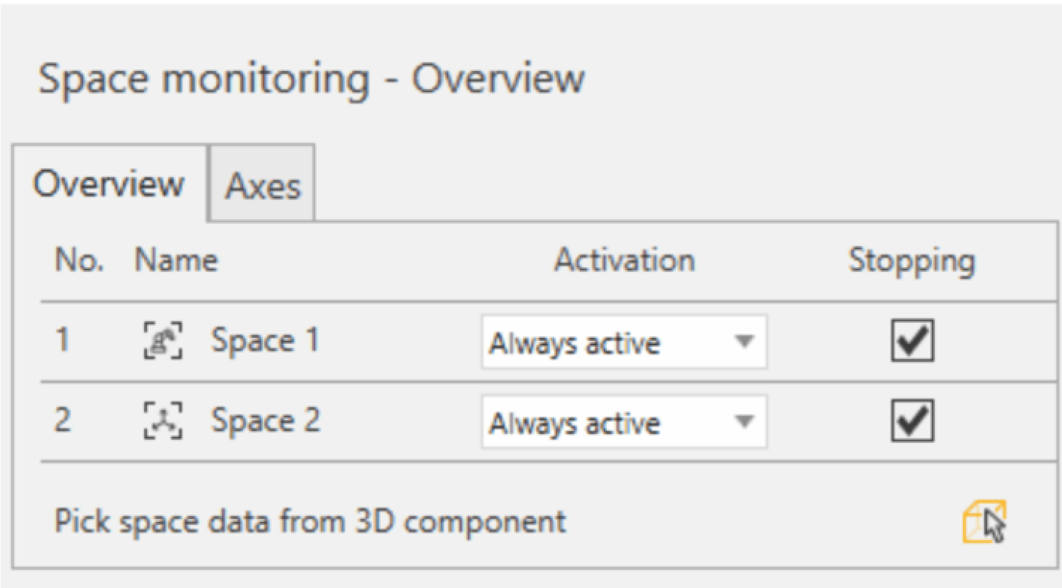
   Some parameters of the Cartesian spaces can also be configured in the 3D scene:

   The shift tool can be used, for example, to grab and shift the Cartesian space with the mouse pointer.

   To use the shift tool, select the following in the ribbon: **Handling** group > **Shift**.

   To place the shift tool on the space, click on the space in the 3D scene.

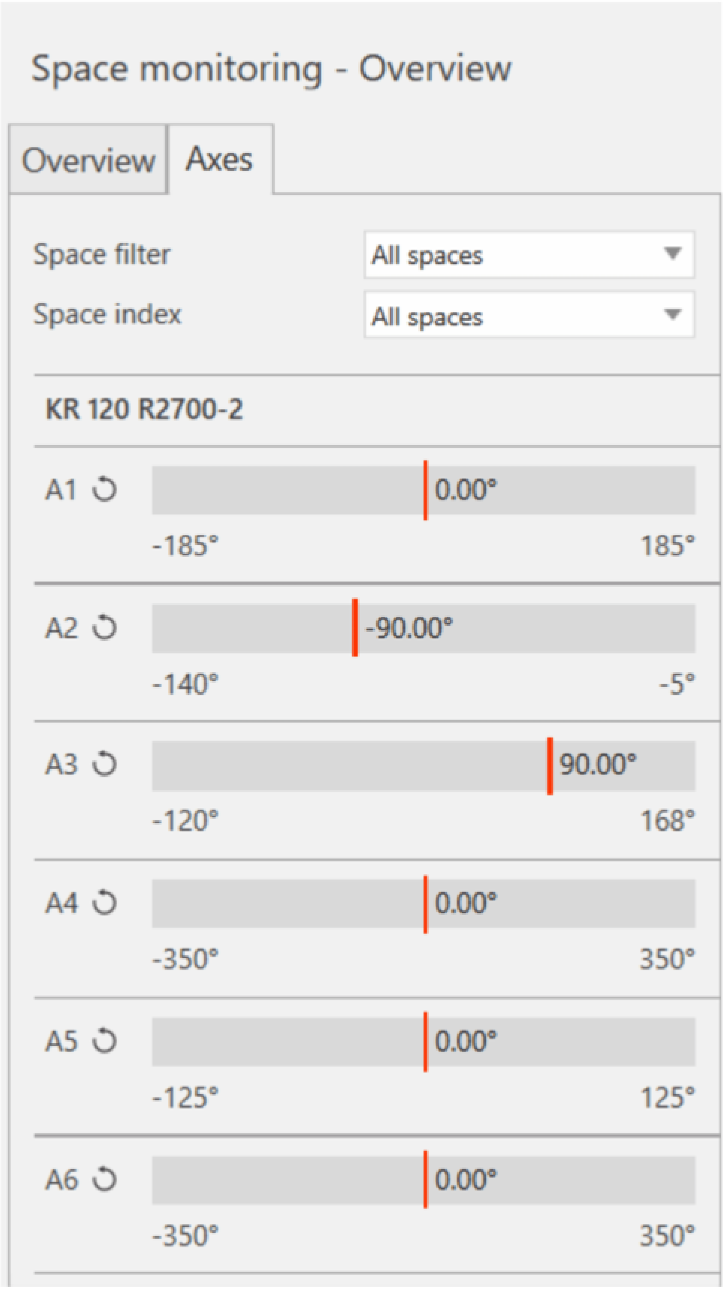
Space monitoring - Overview



Space monitoring - Overview

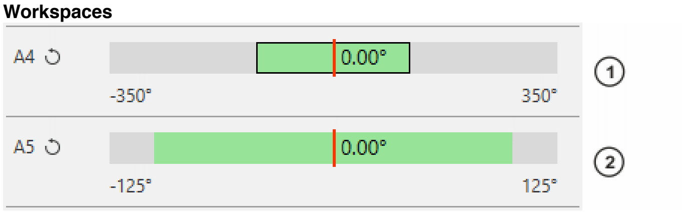
Box	Description
No.	Space number
Name	Space name, editable The icon next to the name indicates the type of monitoring space.
Activation	Activating the monitoring space • **Always active**: Monitoring space is always active. (Default) • **By input**: Monitoring space is activated by a safety-oriented input. • **Inactive**: Monitoring space is not active. If this option is selected, the configurations of the respective monitoring space are lost.
Stopping	A stop is triggered if the space is violated. • Check box active: Robot stops if the monitoring space limits are exceeded. (Default) • Check box not active: Robot does not stop if the monitoring space limits are exceeded.
Pick space data from 3D component	After clicking on the button, a previously created space can be defined as the new monitoring space in the 3D scene. The space is framed in red when selected. (>>> Creating 3D objects as a template for the safety configuration)

Space monitoring – Axes



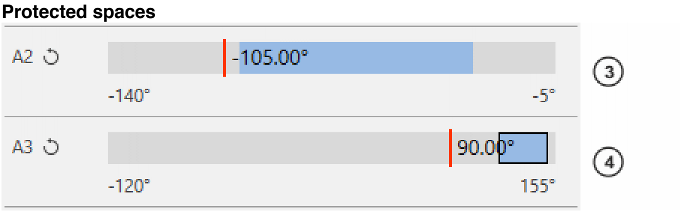
Space monitoring – Axes

Box	Description
Space filter	Space filter • **All spaces**: Displays all configured monitoring spaces in a merged view. Also displays the spaces which are not active until the relevant input is switched. If only 1 space is selected in the space index, only that space will be displayed. • **Active spaces**: Displays all active monitoring spaces. • **Stopping spaces**: Displays all active monitoring spaces which will cause the robot to stop if it violates a space. • **Violated stopping spaces**: Displays all stopping monitoring spaces that are violated. • **All alarm spaces**: Displays all configured alarm spaces, even if they are not active until the relevant input is switched. If only 1 alarm space is selected in the space index, only that alarm space will be displayed. • **Active alarm spaces**: Displays all active alarm spaces.
Space index	Space index • **All spaces**: Displays a merged view of the configured active monitoring spaces as a function of the space filter. If only the alarm spaces are selected in the space filter, only the active alarm spaces will be displayed. • *Number of space* **–** *Name of space*: Displays only the selected configured monitoring space. **Note**: If no axis limits are configured for the selected monitoring space of an axis, that axis will be grayed out.



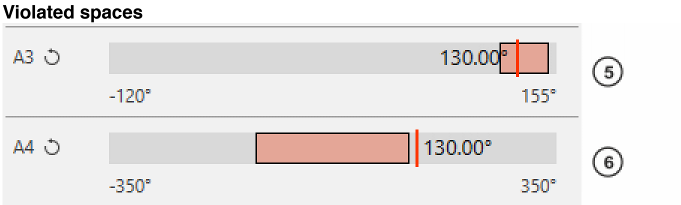
Representation of workspaces

1	Stopping workspace	2	Non-stopping workspace



Representation of protected spaces

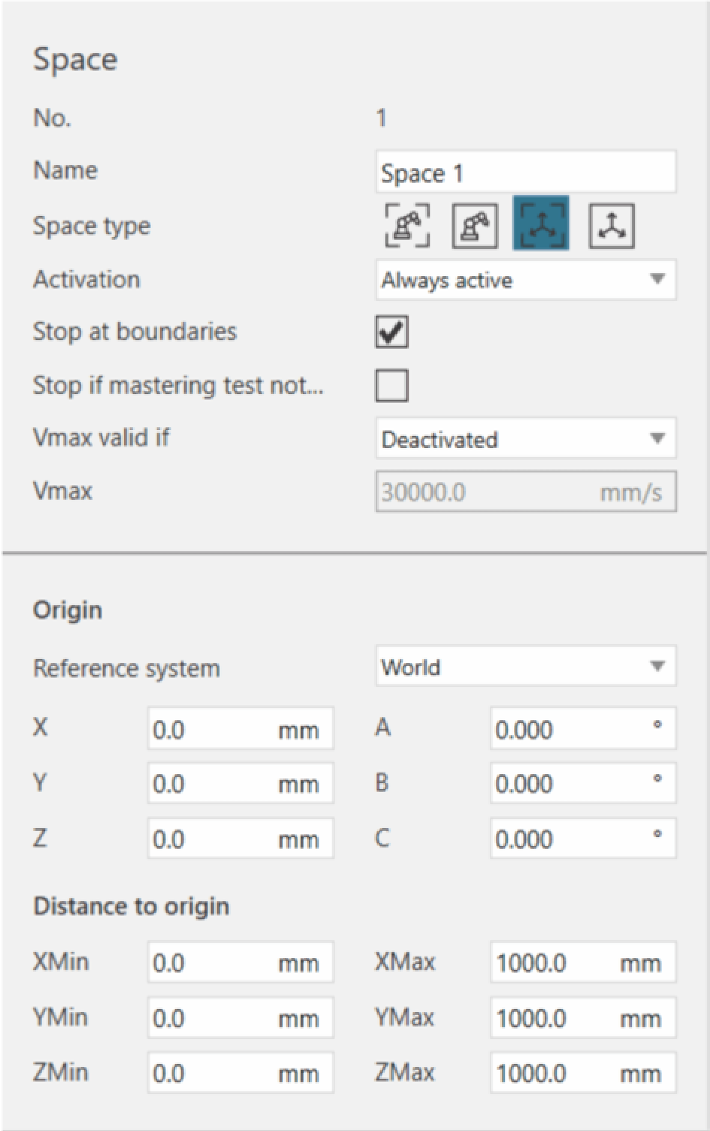
1	Non-stopping protected space	2	Stopping protected space



Representation of violated spaces

5	Violated stopping protected space
6	Violated stopping workspace

Cartesian space



Cartesian space

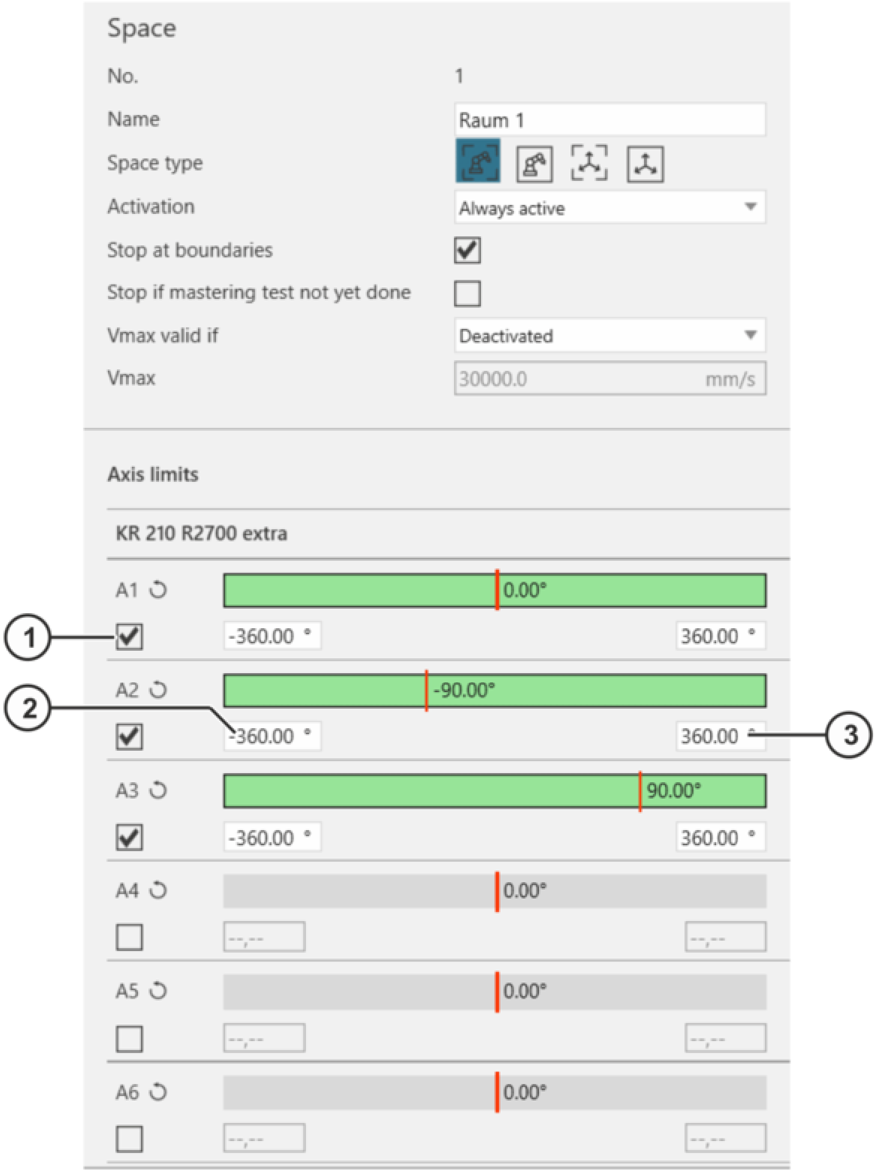
General boxes (identical for Cartesian spaces and axis spaces)

Box	Description
No.	Space number
Name	Space name, editable
Space type	Hovering over the icons with the mouse shows which type of monitoring space they represent. Click on an icon to define the type of monitoring space. • **Axis-specific workspace** • **Axis-specific protected space** • **Cartesian workspace** • **Cartesian protected space** **Workspace** = The safety-oriented tool must move within the configured limits of the monitoring space. (Space violation if the safety-oriented tool leaves the monitoring space.) **Protected space** = The safety-oriented tool must move outside the configured limits of the monitoring space. (Space violation if the safety-oriented tool enters the monitoring space.)
Activation	Activating the monitoring space • **Always active**: Monitoring space is always active. (Default) • **By input**: Monitoring space is activated by a safety-oriented input. • **Inactive**: Monitoring space is not active
Stop at boundaries	A stop is triggered if the space is violated. • Check box active: Robot stops if the monitoring space limits are exceeded. (Default) • Check box not active: Robot does not stop if the monitoring space limits are exceeded.
Stop if mastering test not yet done	Activating reference stop • Check box active: Reference stop is activated for the monitoring space. • Check box not active: Reference stop is not activated for the monitoring space. (Default)
Vmax valid if	Validity of the space-specific velocity • **Deactivated**: Space-specific velocity is not monitored. (Default) • **Space not violated**: Space-specific velocity is monitored if the monitoring space is not violated. • **Space violated**: Space-specific velocity is monitored if the monitoring space is violated.
Vmax	Limit value of the space-specific velocity • **0.5 … 30 000 mm/s** Default: 30 000

Boxes for Cartesian spaces only

Box	Description
Reference system	Reference coordinate system • **World** (default) • **RobRoot**
Origin: X, Y, Z	Offset of the origin of the Cartesian monitoring space in X, Y and Z relative to the selected reference coordinate system. • **-100,000 mm … +100,000 mm** Default: 0 mm
Origin: A, B, C	Orientation in A, B and C at the origin of the Cartesian monitoring space relative to the selected reference coordinate system. Origin A, C: • **-180° … +180°** Origin B: • **-90° … +90°** Default: 0°
Distance to origin: XMin, YMin, ZMin	Minimum X, Y and Z coordinates of the Cartesian monitoring space relative to the origin • **-100,000 mm … +100,000 mm** Default: 0 mm
Distance to origin: XMax, YMax, ZMax	Maximum X, Y and Z coordinates of the Cartesian monitoring space relative to the origin • **-100,000 mm … +100,000 mm** Default: 0 mm

Axis space



Axis space

Axis limit range

Item	Description
1	Activation of monitoring • Check box active: Monitoring is activated. • Check box not active: Monitoring is not activated. (Default)
2	Lower limit of the axis-specific monitoring space (lower axis angle) • Rotational axes: **-360° … +360°** Default: -360° • Linear axes: **-30 000 mm … +30 000 mm** Default: -30 000 mm The lower limit of an axis-specific workspace must be at least 0.5° or 1.5 mm less than the upper limit.
3	Upper limit of the axis-specific monitoring space (upper axis angle) • Rotational axes: **-360° … +360°** Default: 360° • Linear axes: **-30 000 mm … +30 000 mm** Default: 30 000 mm The upper limit of an axis-specific workspace must be at least 0.5° or 1.5 mm greater than the lower limit.

General boxes (identical for Cartesian spaces and axis spaces)

Box	Description
No.	Space number
Name	Space name, editable
Space type	Hovering over the icons with the mouse shows which type of monitoring space they represent. Click on an icon to define the type of monitoring space. • **Axis-specific workspace** • **Axis-specific protected space** • **Cartesian workspace** • **Cartesian protected space** **Workspace** = The safety-oriented tool must move within the configured limits of the monitoring space. (Space violation if the safety-oriented tool leaves the monitoring space.) **Protected space** = The safety-oriented tool must move outside the configured limits of the monitoring space. (Space violation if the safety-oriented tool enters the monitoring space.)
Activation	Activating the monitoring space • **Always active**: Monitoring space is always active. (Default) • **By input**: Monitoring space is activated by a safety-oriented input. • **Inactive**: Monitoring space is not active
Stop at boundaries	A stop is triggered if the space is violated. • Check box active: Robot stops if the monitoring space limits are exceeded. (Default) • Check box not active: Robot does not stop if the monitoring space limits are exceeded.
Stop if mastering test not yet done	Activating reference stop • Check box active: Reference stop is activated for the monitoring space. • Check box not active: Reference stop is not activated for the monitoring space. (Default)
Vmax valid if	Validity of the space-specific velocity • **Deactivated**: Space-specific velocity is not monitored. (Default) • **Space not violated**: Space-specific velocity is monitored if the monitoring space is not violated. • **Space violated**: Space-specific velocity is monitored if the monitoring space is violated.
Vmax	Limit value of the space-specific velocity • **0.5 … 30 000 mm/s** Default: 30 000

Creating 3D objects as a template for the safety configuration

Description

In addition to the possibility of defining monitoring spaces and safety-oriented tools directly in the safety configuration, 3D objects can also be created as templates for this purpose. These can be edited, saved and applied later in the safety configuration.

(>>> Space monitoring - Overview)

(>>> “Tool” and “Spheres”)

Precondition

- Robot present in 3D scene
- KUKA.Sim Modeling add-on is activated
- MODELING area

Procedure

Cartesian safety space

1. In the 3D scene, select the component for which a Cartesian monitoring space is to be created.
2. Under **Features** > **Safety configuration**, click on **Cartesian safety space**. A space is inserted into the 3D scene.
3. Position the space directly in the 3D scene or by entering coordinates in the **Feature Properties** window.
4. If necessary, make further settings in the **Feature Properties** window.

Safety tool sphere

1. In the 3D scene, select the tool that is to be created as a safety-oriented tool.
2. Under **Features** > **Safety configuration**, click on **Safety tool sphere**. A sphere is inserted into the 3D scene.
3. Position the sphere directly in the 3D scene or by entering coordinates in the **Feature Properties** window.

4. If necessary, make further settings in the **Feature Properties** window.

Safety tool TCP

1. In the 3D scene, select the tool that is to be created as a safety-oriented tool.
2. Under **Features** > **Safety configuration**, click on **Safety tool TCP**. A TCP is inserted into the 3D scene.
3. Position the TCP directly in the 3D scene or by entering coordinates in the **Feature Properties** window.

Activating a monitoring space by means of an input

Description

Monitoring spaces can be activated and deactivated by inputs. There are various options for setting inputs for monitoring spaces:

- Using a script
- Manually during the simulation
- Using statements in the program editor

Precondition

- Robot present in 3D scene
- Safe monitoring activated
- Field bus configuration imported
- Monitoring space present
- START area

Procedure

1. In the **Devices** window, click on the desired monitoring space under **Safety configuration** > **Monitoring spaces**. The corresponding settings are displayed in the **Component properties** window.
2. Under **Activation**, click on the selection **By input**.

Setting an input for a monitoring space by means of a script

Precondition

- Robot present in 3D scene
- Safe monitoring activated
- Field bus configuration imported
- Monitoring space present
- START or PROGRAMMING area

Procedure

1. Select the component **Safety PLC** in the 3D scene or in the **Devices** window.
2. Click on **Edit script** in the **Component properties** window. A window opens with a Python script as a template.
3. Edit or expand the Python script as required.

Setting an input for a monitoring space manually

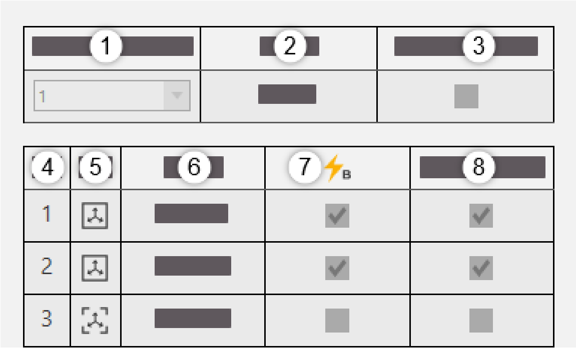
Precondition

- Robot present in 3D scene
- Safe monitoring activated
- Field bus configuration imported
- Monitoring space present
- START or PROGRAMMING area

Procedure

1. Click on the component **Safety PLC** in the 3D scene or in the START > **Devices** area. The **Component properties** window opens.
2. Start the simulation.
3. Activate or deactivate the individual monitoring spaces as required in the **Component properties window**.

Component properties



Setting an input for a monitoring space manually

Item	Description
1	Selection of the safe tool with which “Braking before restricted areas” is to be simulated.
2	Name of the selected tool.
3	• Check box active: All changes to the tool (e.g. by means of scripts or statements) are ignored. The selection remains active even after the simulation has been completed. • Check box not active: Changes to the tool are carried out. The selection is reset again on completion of the simulation.
4	Number of the monitoring space
5	Type of monitoring space
6	Name of the monitoring space
7	• Check box active: Input for the respective monitoring space is active. • Check box not active: Input for the respective monitoring space is not active.
8	• Check box active: All changes to the input (e.g. by means of scripts or statements) for the respective monitoring space are ignored. The selection remains active even after the simulation has been completed. • Check box not active: Changes to the input for the respective monitoring space are carried out. The selection is reset again on completion of the simulation.

Setting an input for a monitoring space by means of a statement

Precondition

- Robot present in 3D scene
- Safe monitoring activated
- Field bus configuration imported
- Monitoring space present
- PROGRAMMING area

Procedure

1. In the program editor, select the line after which the statement is to be inserted.
2. Click on **Logic** > **RCS safety space input**. The statement is inserted into the program editor.
3. Select the number of the desired monitoring space under **Space number**in the **Statement Properties window**.
4. Depending on whether the monitoring space is to be activated or deactivated at this point, activate the check box next to **Input value** or leave it blank.

Configuring the reference position and reference group

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been assigned to the project.

Precondition

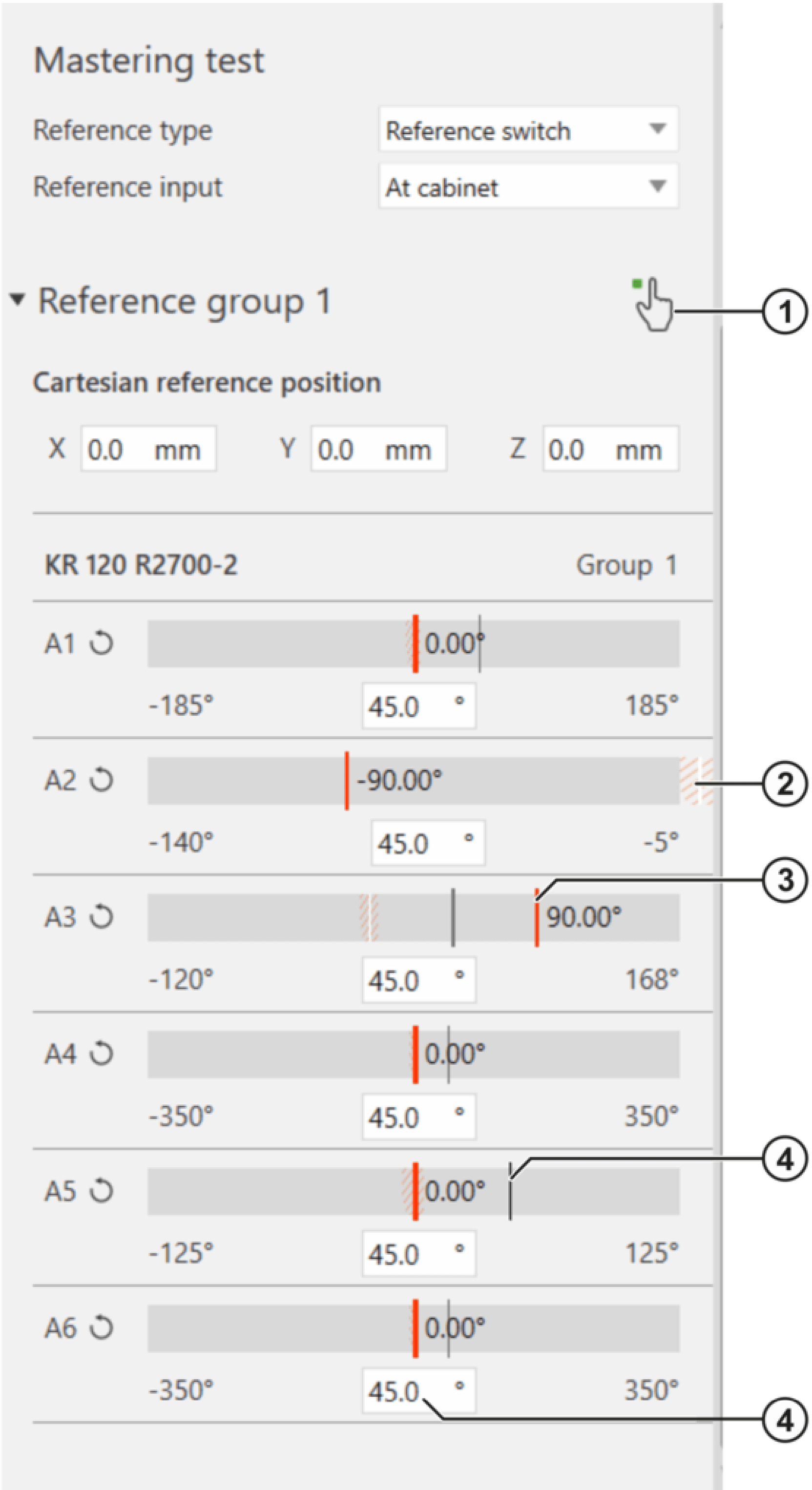
- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Reference position** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.

2. Select the mastering type **Reference switch** in the **Reference type** box.
3. If external axes are added, select for each external axis the number of the reference group to which it is to be assigned.
4. Move all axes of reference group 1 to the corresponding reference position. Use the **Interact** tool for this: Group **Handling** > **Interact**.
5. If present, also jog the external axes to the corresponding reference position.
6. Click on the button to apply the reference position for all axes.

Reference position



Reference position

Item	Description
1	Button for touch-up
2	Range in which the axis must not be located before touch-up
3	Axis-specific actual position
4	Axis-specific coordinates of the reference position To monitor the mastering, the axis angles of the robot axes are defined for a specific Cartesian reference position. During the mastering test, the robot moves to the Cartesian reference position and the actual position of the axes is compared with the setpoint position. • Rotational axes: **-360° … +360°** Default: 45° • Linear axes: **-30 000 mm … +30 000 mm** Default: 1000 mm

Mastering test

Box	Description
Reference type	• **Reference switch**: Mastering test is carried out via KUKA reference switch. (Default) • **External confirmation**: Mastering test is performed via external system and with external mastering confirmation.
Reference input	Mastering type **Reference switch**: • **At cabinet**: Reference switch is connected via interface X42 or XG42. • **Via bus interface**: Reference switch is connected via Ethernet safety interface. Mastering type **External confirmation**: • **At cabinet**: Mastering is confirmed via interface X42 or XG42. (Default) • **Via bus interface**: Mastering is confirmed via Ethernet safety interface.

Reference group 1

Box	Description
Cartesian reference position	X, Y and Z coordinates of the Cartesian reference position relative to the WORLD coordinate system (display for reference group 1) The coordinates of the Cartesian reference position refer to the center point of the mounting flange. • **-30 000 mm … +30 000 mm** Default: 0 mm
Reference group	Each axis that is to be subjected to safe monitoring must be assigned to a reference group. There are 3 reference groups: • Reference group **1** Robot axes and external axes on which a robot is installed are always assigned to reference group 1. Such external axes include KLs, for example, that serve as a carrier kinematic system for the robot. • Reference group **2**, **3** Only external axes can be assigned to reference groups 2 and 3. There must be no robot installed on them. Reference groups **2** and **3** are only available if external axes are present in the 3D scene.

Icons

Icon	Description
↻	Icon for rotational and infinitely rotating axes
→	Icon for linear axes

Configuring axis-specific velocity monitoring functions and braking time

Description

The axis velocity can be monitored against various different limit values:

- Limit value for the reduced axis velocity (activated by means of the safety-oriented input VRED)

  If no safety interface is used, this monitoring function is dispensed with, as the input VRED is not available.
- Limit value for maximum axis velocity in T1

Furthermore, the braking ramp of an axis can be changed using the braking time parameter.

Monitoring of the braking ramp and the maximum axis velocity in T1 is part of the standard safety configuration and always active. The parameters can also be modified if safe monitoring is deactivated.

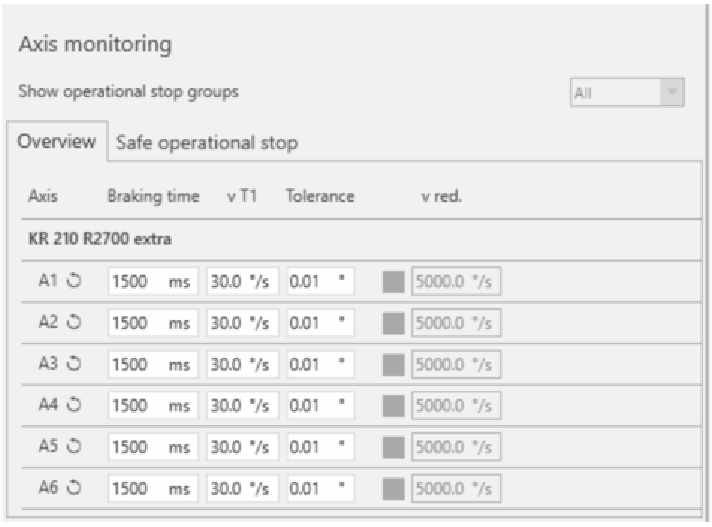
Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Axis monitoring** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.
2. Edit the parameters as desired.
3. If necessary, activate monitoring of the safely reduced axis velocity for one axis. To do so, activate the check box in the row corresponding to the desired axis.
4. Change the limit value for the safely reduced axis velocity (= **v red.**) if necessary.

Axis monitoring – Overview



Axis monitoring – Overview

Parameter	Description
Show operational stop groups	Filter for the axis groups • **All**: All axes are displayed, irrespective of whether or not they are monitored in an axis group. • **1 … 6**: Only the axes that are monitored in the selected axis group are displayed.
Braking time	Duration of the axis-specific braking ramp monitoring for safety stop 1 and safety stop 2 • **500 … 15 000 ms** Default: 1 500 ms
v T1	Maximum axis velocity in T1 • Rotational axes: **1.0 … 100.00°/s** Default: 30°/s • Linear axes: **1.0 … 1 500 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.
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
v red.	Limit value for safely reduced axis velocity • Rotational axes: **0.5 … 5 000 °/s** Default: 5 000°/s • Linear axes: **1.5 … 10 000 mm/s** Default: 10 000 mm/s **Note**: For configuration of this parameter, the option package KUKA.SafeOperation or a test version of this option package is required.

Configuring the safe operational stop for the axis groups

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation or a different safety option has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or another safety option has been assigned to the project.

Description

Safe operational stop can be configured for up to 6 axis groups (situations). A maximum of 8 axes or, in the case of kinematic systems with master/slave axes, a maximum of 8 drives can be configured for each axis group.

The safe operational stop for axis groups is activated via safety-oriented inputs. If none of the safety interfaces specified in the product description is used, the safe operational stop for axis groups is not available.

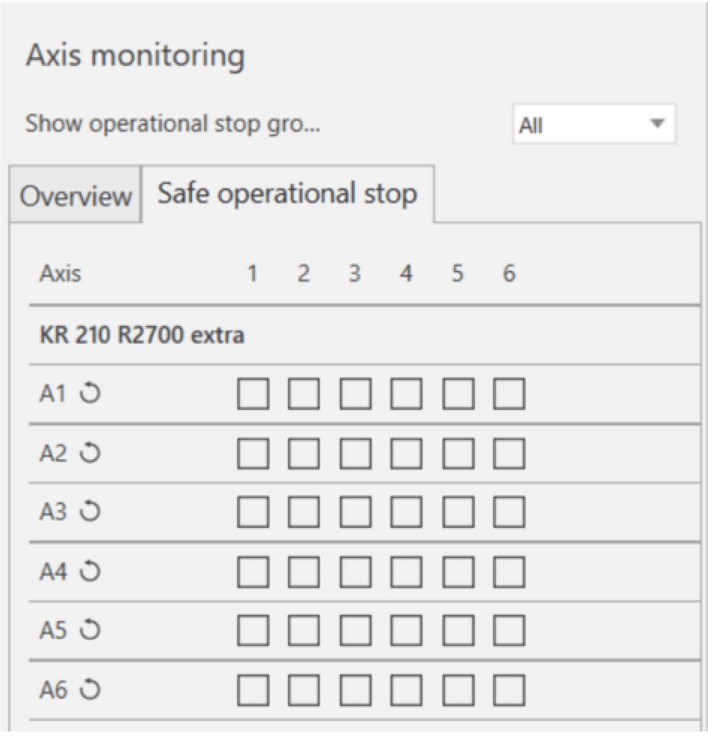
Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Axis monitoring** under **Safety configuration**.
2. Under **Axis monitoring** in the **Properties** window, select the **Safe operational stop** tab.
3. Select axis from the list.
4. Activate one or more axis groups in which the axis is to be monitored by activating the corresponding check box (set the check mark).
5. Repeat steps 3 to 4 to define further monitoring functions.

Axis monitoring – Safe operational stop



Axis monitoring – Safe operational stop

Element	Description
Show operational stop groups	Filter for the axis groups • **All**: All axes are displayed, irrespective of whether or not they are monitored in an axis group. • **1 … 6**: Only the axes that are monitored in the selected axis group are displayed.
Check box	Safe operational stop for axis group 1 … 6 • Check box active: Axis is monitored in axis group. • Check box not active: Axis is not monitored in axis group. (Default)

Configuring velocity monitoring functions

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation or a different safety option has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or another safety option has been assigned to the project.

Description

The Cartesian velocity can be monitored against various different limit values:

- Limit value for maximum global Cartesian velocity (not space-dependent)
- Limit value for safely reduced Cartesian velocity
- Limit value for safely reduced Cartesian velocity in T1 mode

The axis velocity can be monitored against the limit value of the maximum axis velocity (valid globally for each axis).

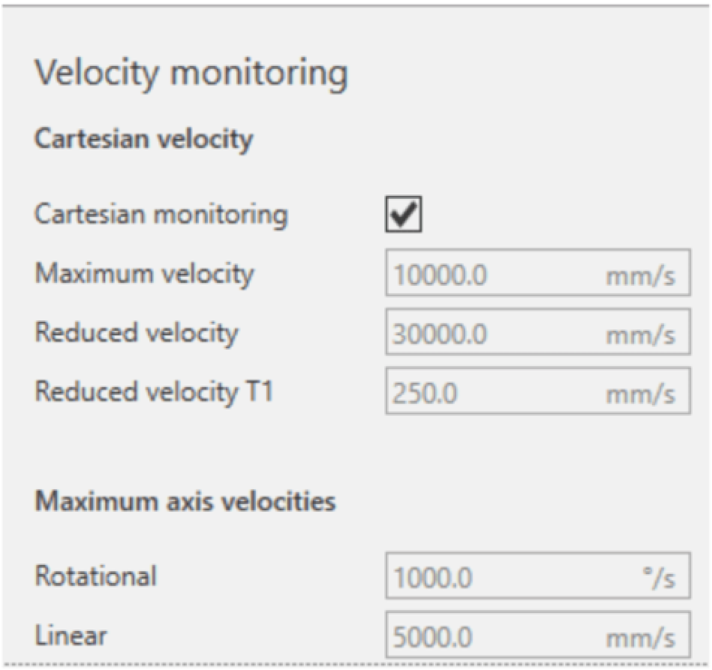
Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Velocity monitoring** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.
2. Edit the parameters as required.

Velocity monitoring



Velocity monitoring

Parameter	Description
Cartesian monitoring	• Check box active: Cartesian monitoring is active. (Default) • Check box not active: Cartesian monitoring is not active.
Maximum velocity	Limit value for global maximum Cartesian velocity (not space-dependent) • **0.5 … 30 000 mm/s** Default: 10 000 mm/s
Reduced velocity	Limit value for safely reduced Cartesian velocity • **0.5 … 30 000 mm/s** Default: 30 000 mm/s
Reduced velocity T1	Limit value for safely reduced Cartesian velocity in T1 mode • **0.5 … 250 mm/s** Default: 250 mm/s
Maximum axis velocities Rotational	Limit value for global maximum velocity for rotational axes • **0.5 … 5 000 °/s** Default: 1 000°/s The axis-specific protected space is dependent on the global maximum axis velocity. A defined minimum size for the axis-specific protected space is derived from the global maximum axis velocity; the size must not fall below this value. If this minimum value is violated, a message is displayed.
Maximum axis velocities Linear	Limit value for global maximum velocity for translational axes • **0.5 … 30 000 mm/s** Default: 5 000 mm/s The axis-specific protected space is dependent on the global maximum axis velocity. A defined minimum size for the axis-specific protected space is derived from the global maximum axis velocity; the size must not fall below this value. If this minimum value is violated, a message is displayed.

Activating “Braking before restricted areas”

Configuration is only possible in the following cases:

- The option package KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been installed and assigned to the project.
- A test version of KUKA.SafeOperation or KUKA.SafeRangeMonitoring has been assigned to the project.

Description

The “Braking before restricted areas” function changes the way the robot behaves at the limits of the monitoring spaces and of the cell area.

With the “Braking before restricted areas” function, the fictitious STOP 1 - DRS end position based on the actual position is monitored in addition to the actual position. If this end point violates a monitoring space, this indicates an impending violation of this monitoring space by the robot.

In this case, a safety stop 1 DRS is triggered, bringing the robot to a standstill at the precalculated end point. Due to the small safety margin in the precalculation, the robot can generally be stopped before the monitoring space.

If the “Braking before restricted areas” function is deactivated, only a velocity reduction to 1% is carried out in the simulation if the monitoring spaces and cell area are touched by a safety-oriented tool. A safety stop is not triggered. A precondition for this is that safe monitoring is activated. The behavior of the robot in the simulation does not correspond to the behavior of the real robot here.

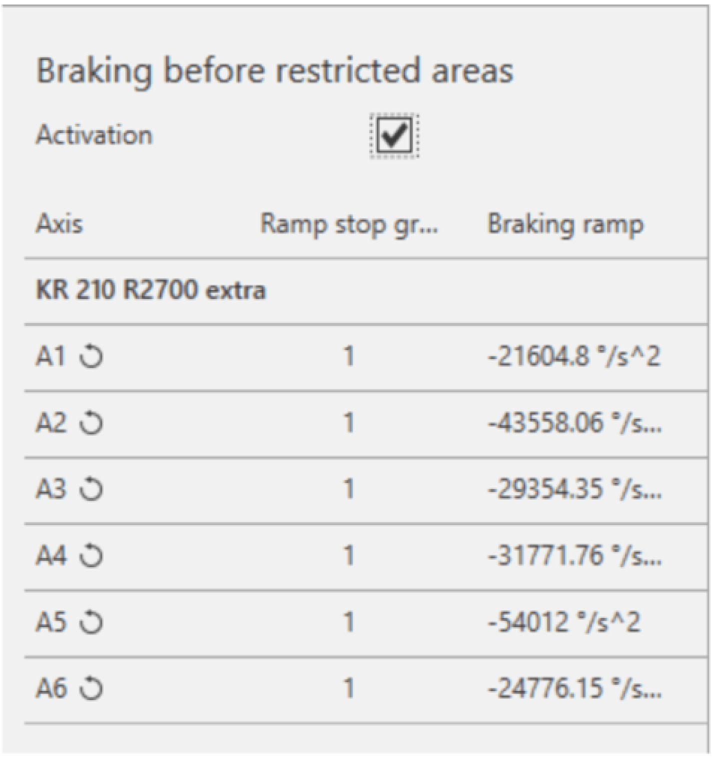
Precondition

- Robot present in 3D scene
- Safe monitoring activated
- START area

Procedure

1. In the **Devices** window, click on **Braking before restricted areas** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.
2. Set the check mark in the check box **Activation**.

Braking before restricted areas



Braking before restricted areas

Parameter	Description
Activation	Activating the function **Braking before restricted areas** • Check box active: Function is activated. • Check box not active: Function is not activated. (Default)
Ramp stop group	Synchronously braking axes belong to a drive ramp stop group (display only)
Braking ramp	Maximum possible braking ramp for an axis (display only)

Icons

Icon	Description
↻	Icon for rotational and infinitely rotating axes
→	Icon for linear axes

Configure hardware options

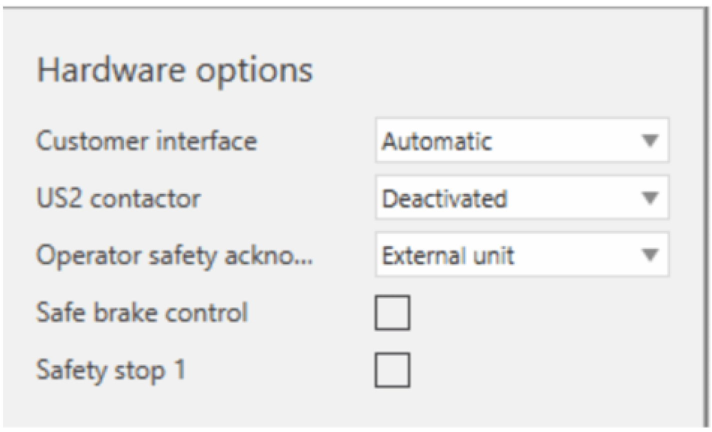
Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Hardware options** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.
2. Carry out the settings as required.

Hardware options



Hardware options

Parameter	Description
Customer interface	Select here which interface is used: • **Automatic** (default) • **Discrete with operating mode output**

US2 contactor	• **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 if the following conditions are met:
	  ◦ No operator safety message is active.
	  ◦ No E2/E7 message is active. (Only relevant for VSS)
	  ◦ Drives are switched on.
	  ◦ The motion enable signal is present.

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. • **Acknowledgement button**: Acknowledgement is given, for example, 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. (Default)
Safe brake control	• Check box active: Brake voltage for the 2nd brake is safely switched off via the output at interface X22. • Check box not active: Brake voltage for the 2nd brake is not safely switched off. (Default)
Safety STOP 1 at input X25	• Check box active: Safety STOP 1 is active via the safe input on interface X25. • Check box not active: Safety STOP 1 is not active. (Default)

Configure the communications parameters

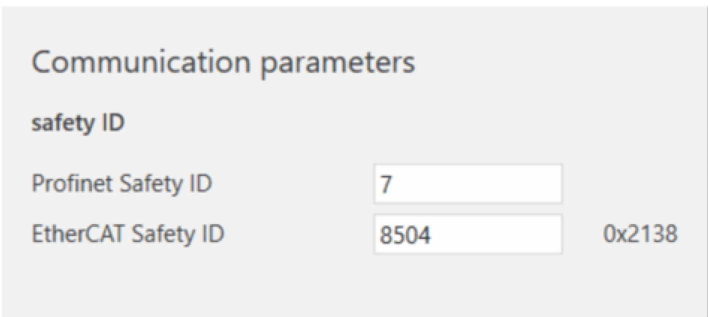
Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Communication parameters** under **Safety configuration**. The corresponding settings are displayed in the **Properties** window.
2. Carry out the settings as required.

Communication parameters



Communication parameters

Parameter	Description
Profinet Safety ID	This ID is required if the robot controller is used as a PROFINET device.
EtherCAT Safety ID	This ID is required if the robot controller is used as an FSoE slave.

Configuring the brake test

Precondition

- Robot present in 3D scene
- START area

Procedure

1. In the **Devices** window, click on **Brake test** under **Safety configuration**. The corresponding settings are displayed in the **Brake test configuration** window.
2. Carry out the settings as required.

Brake test configuration

Parameter	Description
Force brake test	• Check box active: The brake test is active. • Check box not active: 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.
Cycle time	The cycle time specifies the interval in hours at which the brake test is to be executed.
Axes for brake test	The robot axes and external axes for which the brake test is to be executed can be selected here.
	Defines the current position of the robot as the start position for the brake test.

RoboTeam

Creating a RoboTeam

Description

There are several ways of creating a RoboTeam in KUKA.Sim:

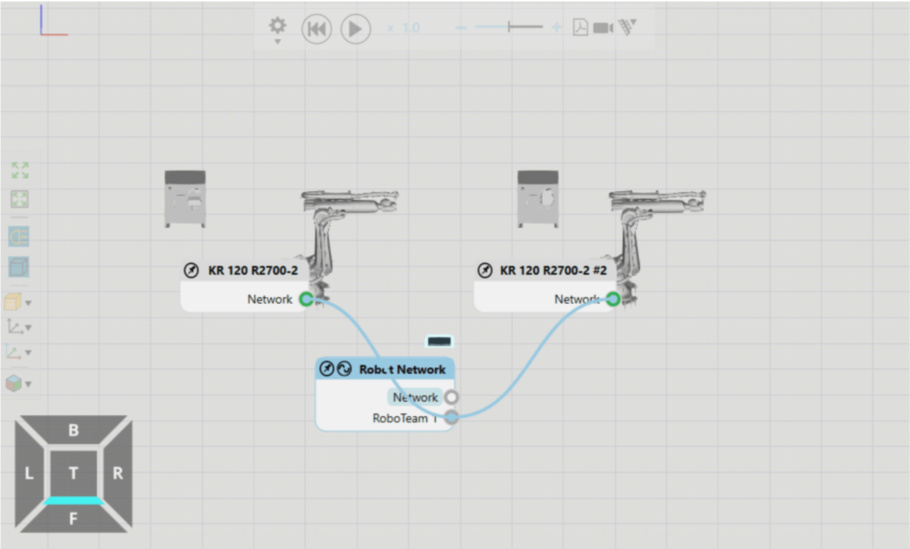
- Using a component from the **eCatalog** window
- Via the **Devices** window
- Via the **Component properties** window of the robot

Precondition

- At least 2 robots present in 3D scene
- START area

Component procedure

1. In the **eCatalog** window, expand the collection **Public Models** > **KUKA.Sim Library** > **KUKA** > **Controllers**.
2. Drag the **Robot Network** component into the 3D scene.
3. Select the component in the 3D scene.
4. Click on the **Add RoboTeam** button in the **Component properties** window.
5. Select ribbon > **Connect** group > **Interfaces**.
6. Drag the **Network** node of the robot to the **RoboTeam** node of the **Robot Network** component to connect it.



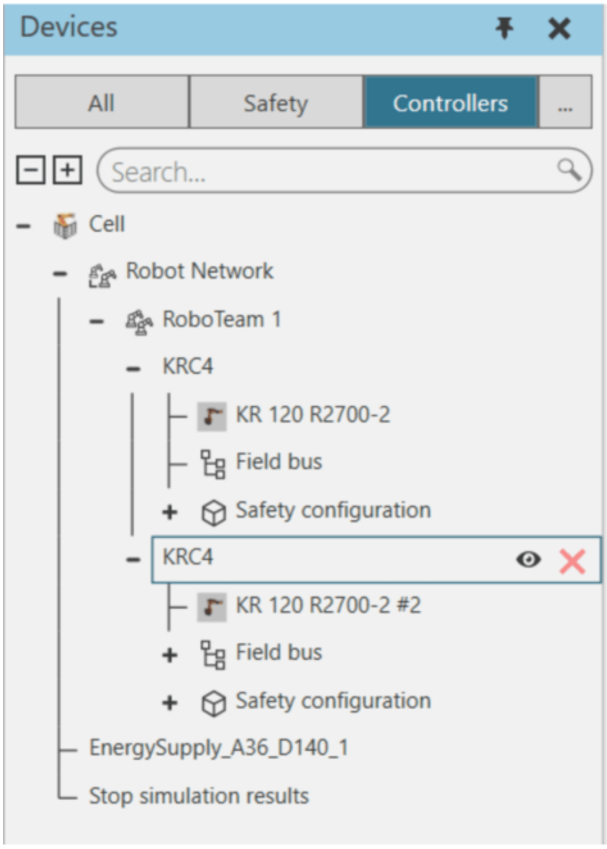
RoboTeam interfaces

7. Repeat step 6 for the other robots that are to be added to the RoboTeam.

Procedure using the Devices window

1. Right-click on the **Cell** node in the **Devices** window and select **Add RoboTeam**. The nodes **Robot Network** and **RoboTeam 1** are added to the device tree.

2. Drag the robot controllers that are to be added to the RoboTeam to the node **RoboTeam 1**.



RoboTeam in the **Devices** window

Procedure using "Component properties" window

1. Select a robot in the 3D scene.
2. Select the **Controller** tab in the **Component properties** window.
3. Select the option **New RoboTeam** in the **Network** box. A RoboTeam with the name **RoboTeam 1** is created and can be selected in the **Network** box.
4. In the 3D scene, select the next robot that is to be added to the RoboTeam.
5. Select **RoboTeam 1** in the **Network** box. The robot is assigned to RoboTeam 1.
6. If further robots are to be added to the RoboTeam, carry out steps 4 and 5 for these robots.

Configuring a RoboTeam

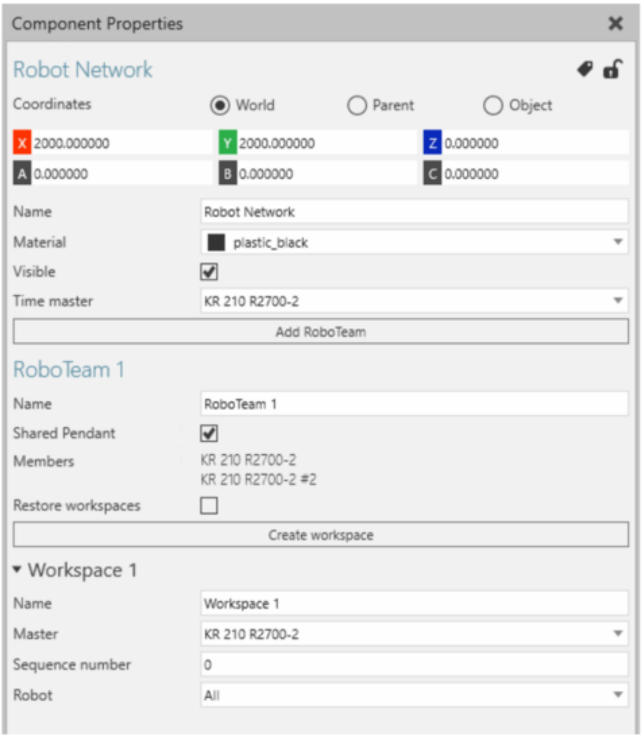
Precondition

- A RoboTeam has been created
- START area

Procedure

1. Click on the component **Robot Network** in the 3D scene.
2. The corresponding settings are displayed under **RoboTeam {No.}** in the **Component properties** window.
3. If a new workspace is to be created, click on **Create workspace**. The settings for this workspace are displayed.
4. Make the desired settings.

Component properties window



RoboTeam and workspace settings

Parameter	Description
Robot Network	
Time master	Select the robot that is to specify the time in the RoboTeam.
RoboTeam{No.}	
Shared Pendant	• Check box active: A shared smartPAD is used in the RoboTeam. • Check box not active: Each RoboTeam participant uses its own smartPAD.
Restore workspaces	• Check box active: If the robot controller is rebooted after a loss of connection, the status of the workspaces can be restored. • Check box not active: The status of the workspaces is not restored.
Workspace {No.}	
Master	Select the robot that manages access to the shared workspace.
Sequence number	Assign a number to the workspace.
Robot	Select which robots are to be granted access to the shared workspace. Only robots for which access is possible are available for selection.

Establishing a geometric coupling

Description

For process-dependent mode with LIN and CIRC motions, the geometric coupling can be established directly in the motion instruction. In KUKA.Sim, the flange of the master robot is selected on the slave robot as the base.

> **i** Further information about process-dependent jogging and geometric coupling can be found in the documentation of the KUKA.RoboTeam option package.

Precondition

- At least 2 robots present in 3D scene
- RoboTeam has been created and configured
- PROGRAM area

Procedure

1. Select the slave robot in the 3D scene.
2. On the **Jog** tab, click on the cogwheel icon next to the **Base** box.
3. Select the flange of the master robot in the **Node** box; there are 2 options for this:
    - Click on the  icon next to the **Node** box and select the flange of the master robot in the 3D scene.
    - Select axis 6 of the master robot in the **Node** box (e.g. for a KR 16-2 this is the entry **KR 16-2::A6**).

The geometric coupling is now established. When the master robot is moved, the slave robot automatically moves with it.

Programming RoboTeam commands

Description

The commands familiar from the option package KUKA.RoboTeam are also available in KUKA.Sim.

Precondition

- At least 2 robots present in 3D scene
- RoboTeam has been created and configured
- PROGRAM area

Procedure

1. Select a robot in the 3D scene.
2. Select in the program area the line after which the command is to be inserted.
3. In the icon bar, select **Logic** > **RoboTeam** and the desired RoboTeam command. The command is inserted; the corresponding settings are displayed in the **Statement Properties** window.
4. Carry out the desired settings in the **Statement Properties** configuration window.

> Further information about the commands can be found in the documentation of the option package KUKA.RoboTeam.

Synchronizing motion blocks

Description

The motions of several robots can be synchronized so that each robot requires the same amount of time for these motions.

> The synchronization of PTP and SPTP motions is only possible if the KUKA.RoboTeam option package has been installed and assigned to the project.

Precondition

- At least 2 robots present in 3D scene
- RoboTeam has been created and configured
- PROGRAM area

Procedure

1. Teach a number of motions on the robots.
2. In the 3D scene, select the robot whose motion is to be synchronized with other robots.
3. In the program area, select the line to be synchronized.
4. Activate the check box next to **Sync motion** in the **Motion sync** area in the **Statement Properties** window.
5. Enter a name for the synchronization flag in the **Label** box.

   > An unambiguous name ensures that only motion blocks of this name in the affected programs are synchronized.

6. Select the synchronization partners in the **Members** box.
7. Carry out steps 3 to 6 for the synchronization partners.

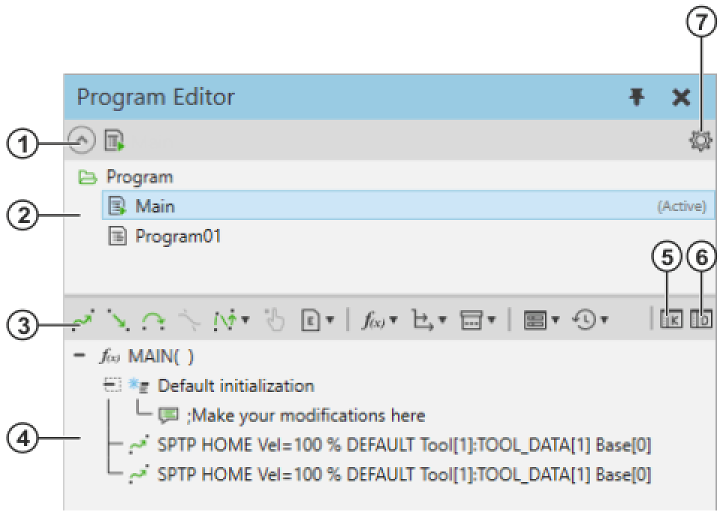
“**Program editor**” window

> Simulation programs do not correspond exactly to reality. Robot programs created in simulation programs must be tested in the system in **Manual Reduced Velocity mode (T1)**. It may be necessary to modify the program.

Overview

The **Program editor** window is displayed by default in the PROGRAM area. It can also be included in the view of other areas.

If a robot is selected in the 3D scene, the **Program editor** window shows its programs.

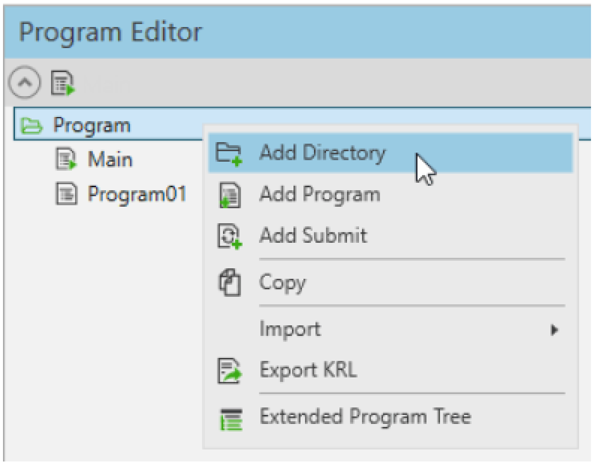


"**Program editor**" window

Item	Description
1	Click to open and close the folder view
2	Open folder view A small green arrow on the program icon indicates which program is selected. This program is executed when simulation is started.
3	**Programming** icon bar (>>> Programming icon bar / inserting commands into program)
4	Program area, here with statement view (>>> "Statement Properties" window)
5	Click to toggle between statement view and KRL view in the program area
6	Click to toggle between SRC view and DAT view in the program area
7	Opens the **Simulation configuration** window. (>>> "Simulation configuration" window)

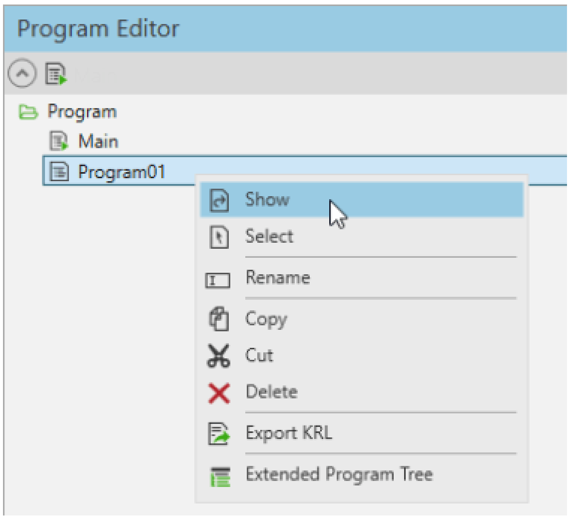
Folder view

Numerous context menu functions are available in the folder view.



Folder view – context menu of the parent level "Program"

Import: KRL directories or files can be imported.



Folder view – context menu for single program

Programming icon bar / inserting commands into program

Inserting a command via the bar

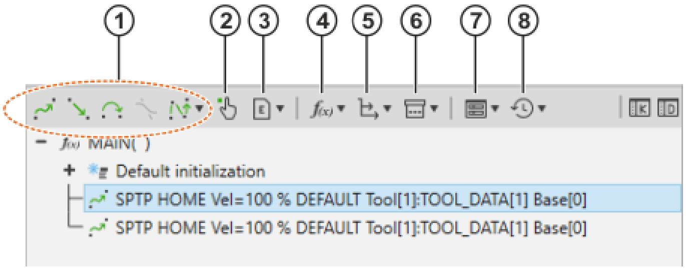
Commands can be inserted into the program via the bar. The commands are familiar from the KUKA System Software.

For motion or logic commands:

- First select in the program area the line after which the new command is to be inserted.

For touch-up:

- First select in the program area the motion command for which the touch-up is to apply.



Programming icon bar

Item	**Description**
1	Insert motion command
2	Perform touch-up for the selected motion command The touch-up applies the position of the robot in the 3D scene.
3	Insert expert command Advanced setting options are available in the **Statement Properties** window for expert commands.
4	Insert function or function call
5	Insert logic command: Initialization, declaration or assignment Loop, control structure, interrupt, I/O-related command or CONTINUE
6	Insert Comment, Print or Fold Insert HALT, WAIT SEC or TIMER **Print**: Enables a text to be displayed in the output window in KUKA.Sim. This command is ignored on the robot controller.
7	Insert inline forms: TRACE, WAIT FOR, TRIGGER, ANALOGSTATIC, ANALOGDYNAMIC Commands for option packages, if used
8	Recent commands

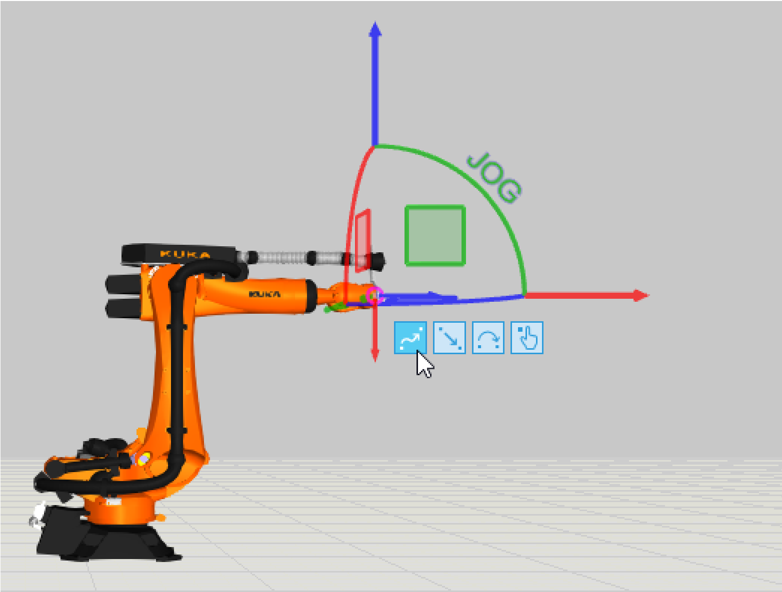
Inserting a motion command via 3D scene

Motion commands can also be inserted into the program via the 3D scene. The current position of the TCP is added to the program as a new point.

- Hovering over an icon in the 3D scene with the mouse displays the name of the command.
- If a line is selected in the program editor, the new command is inserted after it.

  If no line is selected, the new command is inserted at the end of the program.

A touch-up can also be performed via the 3D scene. The touch-up (hand icon) is only available if a motion command is selected in the program editor.

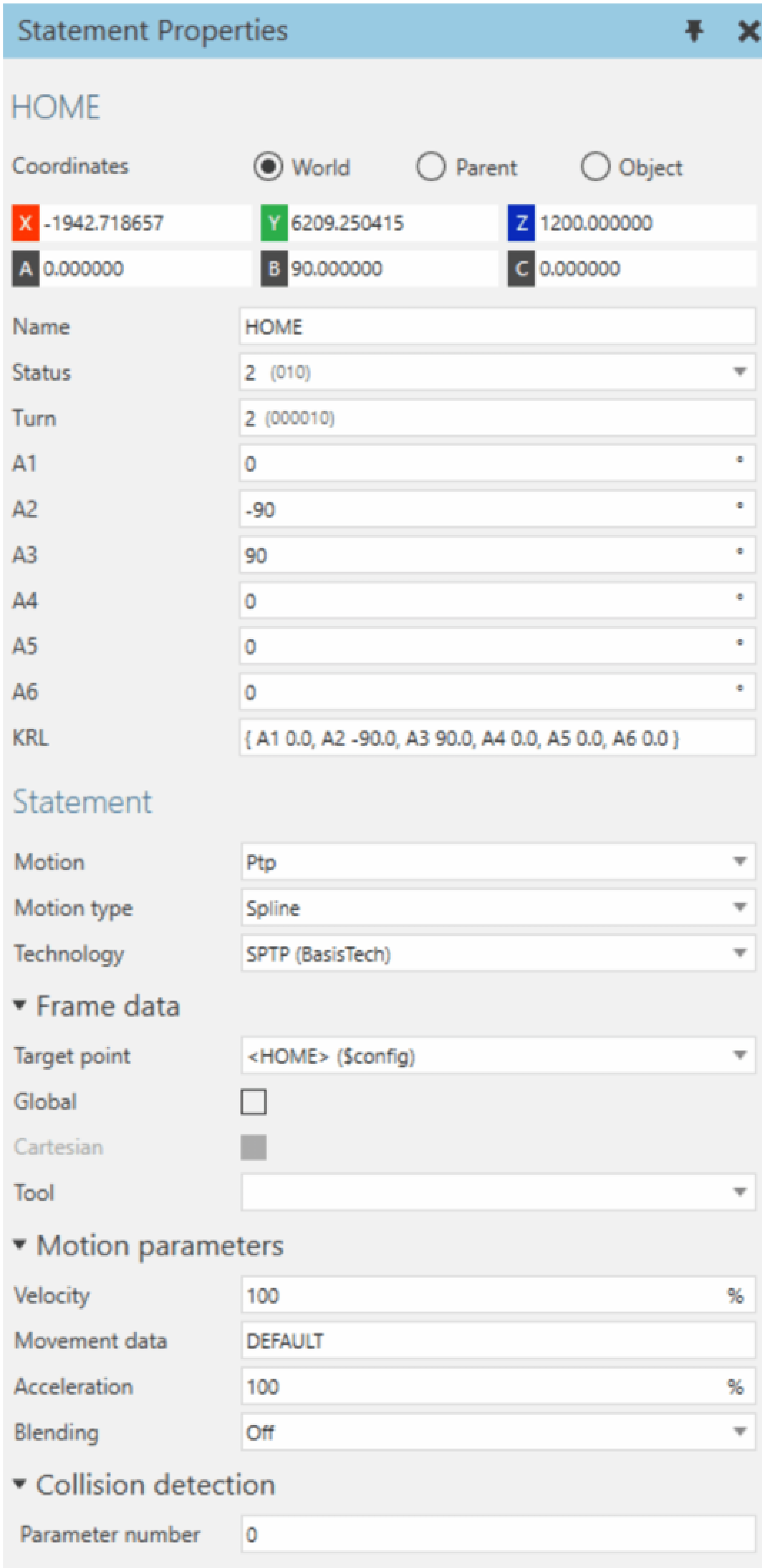


Inserting a motion command via 3D scene

“**Statement Properties**” window

Double-clicking on a statement in the program area opens the **Statement Properties** window. The properties are specific to each statement.

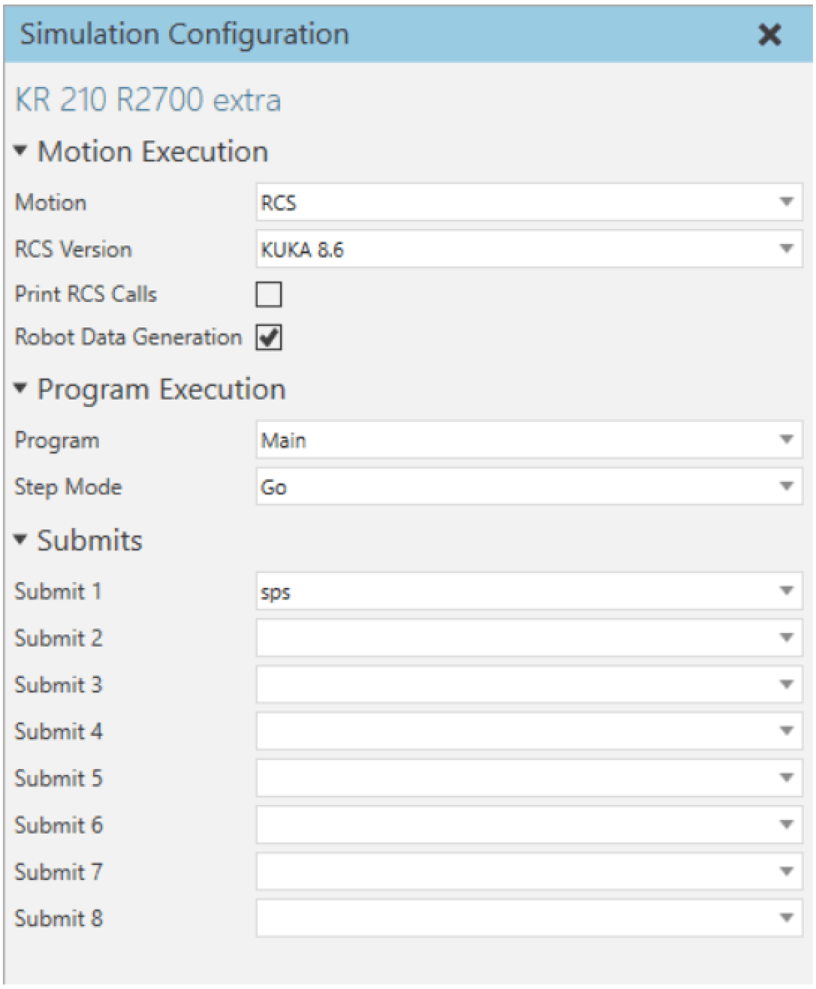
Here, for example, are the properties for a motion statement. The properties are familiar from the KUKA System Software.



"**Statement Properties**" window (example: motion instruction)

Box	**Description**
Motion type	• **Classic**: Conventional PTP, LIN and CIRC motions are used for programming. • **Spline**: Spline motions are used for programming.
Technology	If option packages are installed, technology-specific entries are available in this box. Selecting the technology-specific entry makes technology-specific commands or parameters available.

“**Simulation configuration**” window



“**Simulation configuration**” window

Box	Description
Motion execution	
Motion	Type of motion execution • **Controller**: **Execution on the connected controller (Office Lite, Office or robot controller)** • **RCS**: **Motion execution with the Robot Controller Simulation (RCS)** • **Integrated**: **Motion and program execution with the built-in KRL interpreter**
RCS version	Here, a different controller version can be selected for the simulation than the one actually assigned to the robot. This makes it possible to consider the behavior of the robot in relation to different controller versions. Actual assignment: (>>> Specifying the supply voltage and controller version)
Print RCS calls	Only for RCS: Activate check box if desired.
Machine data generation	Only for RCS: Activate check box if desired.
Program	Select program to be executed.
Step mode	Select step mode.
Submits	If required: assign further submit programs

Information about the step modes can be found under “Program run modes” in the Operating and Programming Instructions for the KUKA System Software.

Information about the submit interpreters can be found in the Operating and Programming Instructions for System Integrators for the KUKA System Software.

“**Monitoring window**” window

Description

The **Monitoring window** window is displayed by default in the PROGRAM area. The state of variables can be monitored in this window. Mathematical calculations (e.g. variable + 5) and expressions as well as nested structures and arrays are also evaluated.

The following variables can be monitored:

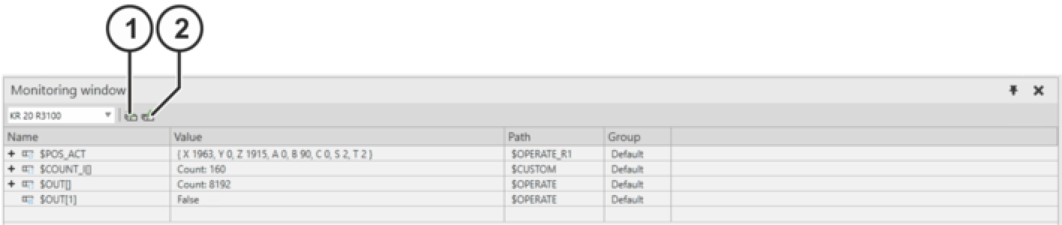
- Variables that have been declared globally (in the file $config.dat or in a DAT file containing the keyword PUBLIC)
- Variables that have been declared locally in the DAT file (globally for the module)
- Variables that have been declared locally in the SRC file while the advance run pointer is located in the same program as the declaration (runtime data)

Precondition

- Robot present in 3D scene
- PROGRAM area

Procedure

- 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.
- Alternatively: Switch to KRL view in the program area. Right-click on the desired variable and select **Monitor**.



“**Monitoring window**” window

Item	Description
1	Importing variables in ConfigMon.ini format
2	Exporting variables in ConfigMon.ini format

Python scripts

Integrating Python scripts

Description

Python scripts can be integrated into KUKA.Sim, e.g. for creating programs automatically.

A detailed description of the possibilities offered by the Python scripts and how these are programmed can be found in the corresponding documentation (HELP area > Python API).

A library containing the KUKA statements can be found as standard in the following directory: C:\Programme\KUKA\KUKA.Sim 4.3\Python\lib\kuka.py

Precondition

- KUKA.Sim is being not executed.

Procedure

1. Create a PY file with the desired scripts.
2. Copy the file into the following directory: C:\Users*Name*\Documents\KUKA\KUKA.Sim 4.3\My Commands
3. Start KUKA.Sim.
4. Switch to the PROGRAM area. The ribbon now contains a group with Python scripts.

Calling Python script for a component

Description

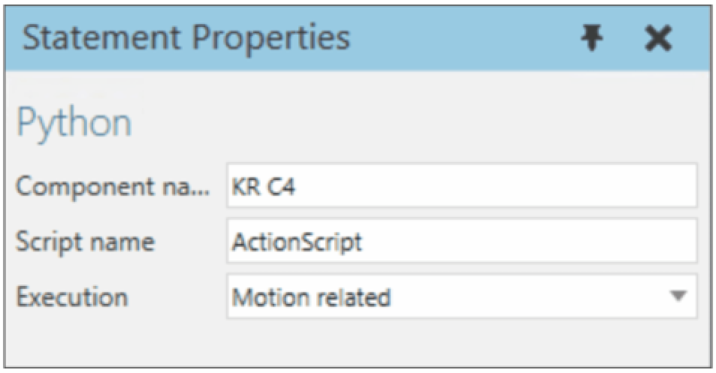
In KUKA.Sim, Python scripts for components can be called in the 3D scene. In this way, certain actions can be carried out using scripts during program execution.

Precondition

- Robot present in 3D scene
- PROGRAM area

Procedure

1. Select the robot in the 3D scene.
2. In the program area, select the line after which the Python script is to be inserted.
3. Open the **Miscellaneous** icon in the icon bar and select **Python**. The script is inserted, the **Statement Properties** window opens on the right-hand side.
4. Carry out the desired settings in the **Statement Properties** configuration window.



Statement Properties window (Python script)

Parameter	Description
Component name	Select the component for which the script is to be called.
Script name	Enter the name of the script.
Execution	• **Interpreter related**: The script is executed when the advance run reaches it. • **Motion related**: The script is executed when the main run reaches it.

Viewing and editing programs online

During the installation of KUKA.Sim, rules for connecting to a KUKA.OfficeLite environment are configured in the firewall. These rules are necessary for KUKA.Sim to be able to establish the connection. Rules are configured for the following applications:

- KUKA.OfficeLite KUKA.Sim application
- KUKA.OfficeLite PortMap service

When KUKA.Sim is uninstalled, these rules are removed again.

Viewing a motion program online in KUKA.Sim

Description

The programs present on a robot controller can be viewed with KUKA.Sim. The behavior of the robot can be observed.

The **View only** mode is used for this purpose. The programs cannot be modified in this mode in KUKA.Sim.

The robot controller can be a KUKA.OfficeLite environment or a real controller. Where OfficeLite is referred to below, it applies to both.

Precondition

- The robot type in the Sim project corresponds to that in OfficeLite.
- The controller version in the Sim project corresponds to that in OfficeLite.

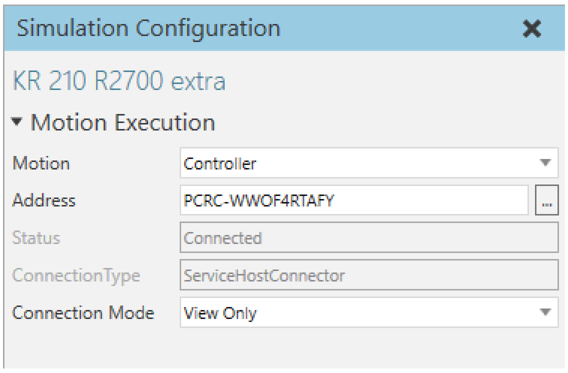
Procedure

1. PROGRAM area > **Program editor** window
2. Click on the cogwheel icon in the top right-hand corner. (If not available, select the robot in the 3D scene.)

   The **Simulation configuration** window opens.
3. Fill out the boxes:
   - **Motion**: **Controller**
   - **Address**:

     Either enter directly the IP address of the controller to which you want to connect.

     Or click on “...”. A window opens. It displays all available controllers. Select the required controller. Confirm with **OK**.
   - **Connection mode**: **View only**



Simulation configuration with "**View only**"

The following occurs:

- The connection to the controller is established. The **Status** box changes to **Connected**.
- A new area appears in the ribbon: **CONNECTION**.

  (>>> CONNECTION ribbon)
- All programs from Office.Lite are displayed in the program editor.

4. Run the program in Office.Lite.

   The robot executes the motions in the 3D scene.

   It does not matter which program is selected in the program editor in KUKA.Sim. The motions of the program running in Office.Lite are displayed.

5. If necessary, edit the program on the controller and/or create additional programs.

   The changes are displayed immediately in KUKA.Sim.

Editing a motion program online in KUKA.Sim

Description

The programs available on a robot controller can be loaded, viewed and modified. The behavior of the robot can be observed in KUKA.Sim. The modified programs can be transferred to a robot controller.

The **Synchronizing** mode is used for this purpose.

The robot controller can be a KUKA.OfficeLite environment or a real controller. Where OfficeLite is referred to below, it applies to both.

Precondition

- The robot type in the Sim project corresponds to that in OfficeLite.
- The controller version in the Sim project corresponds to that in OfficeLite.

Preparation

1. Export the project from KUKA.Sim.
2. Open and save the project in WorkVisual.
3. Transfer the project from WorkVisual to OfficeLite, retaining the existing programs there.

Preparation is not mandatory, but is recommended. It consolidates the set of data and ensures that there are fewer differences and conflicts to process in the transfers described below.

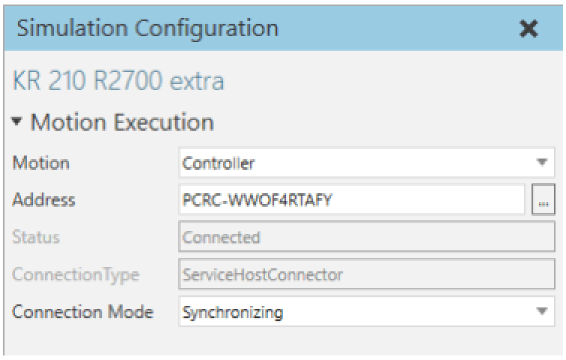
Procedure

1. PROGRAM area > **Program editor** window
2. Click on the cogwheel icon in the top right-hand corner. (If not available, select the robot in the 3D scene.)

   The **Simulation configuration** window opens.
3. Fill out the boxes:
   - **Motion**: **Controller**
   - **Address**:

     Either enter directly the IP address of the controller to which you want to connect.

     Or click on "...". A window opens. It displays all available controllers. Select the required controller. Confirm with **OK**.
   - **Connection mode**: **Synchronizing**



Simulation configuration with "**Synchronizing**"

The following occurs:

- The connection to the controller is established. The **Status** box changes to **Connected**.
- A new area appears in the ribbon: **CONNECTION**.

  (>>> CONNECTION ribbon)

4. If desired, run a program in Office.Lite.

   The robot executes the motions in the 3D scene.

   It is not relevant which program is selected in the program editor in KUKA.Sim. The motions of the program running in Office.Lite are displayed.

5. Load files from the controller:

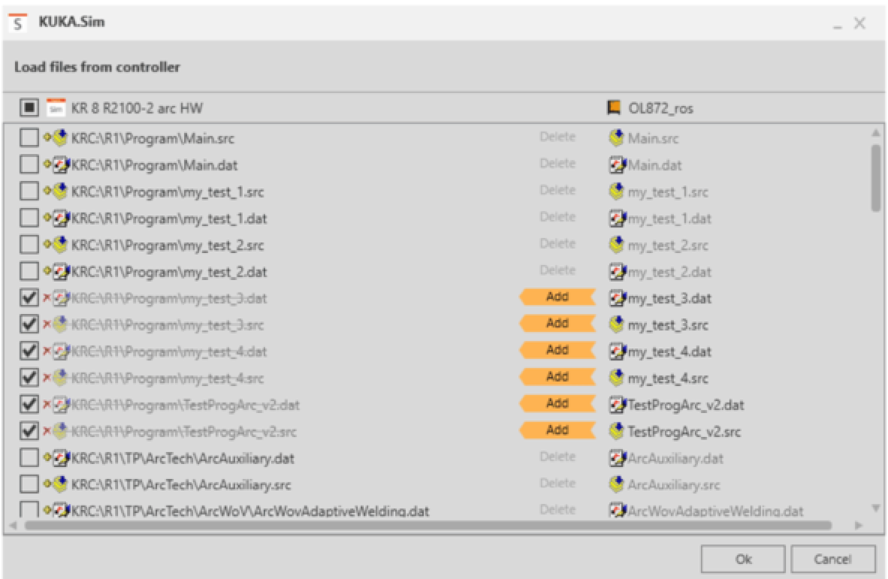
   - Either load all files:

     CONNECTION ribbon > **Download programs from controller**
   - Or load only the files of certain programs:

     Select one or more programs in the program editor > right-click > **Load files from controller**
   - Or load only the active program via the program editor:

An overview opens: It shows the differences between the Sim project and OfficeLite.

A check mark indicates which adaptations would be made in the Sim project. However, the user can decide individually for each difference which state he wishes to have in the Sim project.



Overview of "**Load files from controller**"

Explanation of the settings using examples from (>>> Overview of "Load files from controller"):

File	Description
cell.src	No check mark preset, as identical in KUKA.Sim and in Office.Lite. **Overwrite** action: grayed out, as no check mark If the user activates the check box, the cell.src in KUKA.Sim will be overwritten with the (identical) cell.src from Office.Lite.
Main.src	No check mark preset, as present in KUKA.Sim and not in Office.Lite (therefore grayed out in right-hand column). **Delete** action: grayed out, as no check mark If the user activates the check box, the Main.src in KUKA.Sim will be deleted.
my_test_1.src	Check mark preset, as not present in KUKA.Sim, but present in Office.Lite (therefore struck through in left-hand column). **Add** action: highlighted, as check box activated If the user leaves the check box activated, the file will be added in KUKA.Sim. If the user deactivates the check box, the file will not be added in KUKA.Sim.

6. Confirm the settings in the overview with **OK**.

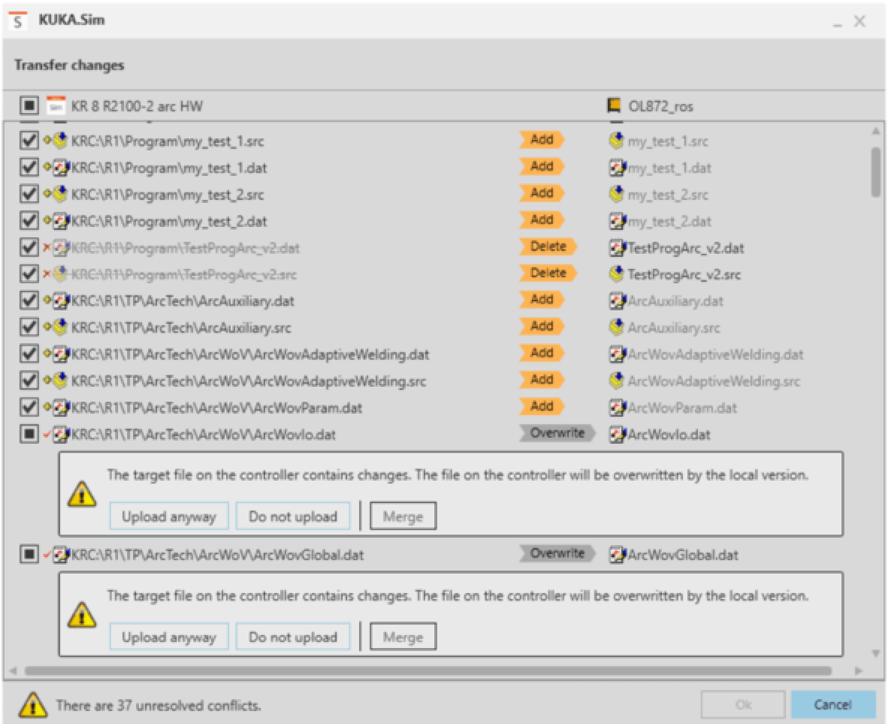
   The selected data are transferred to KUKA.Sim. In the overview, a green check mark appears on the right-hand side of the lines that have already been processed. The program editor displays the transferred programs.

   The overview closes automatically when the transfer has been completed.
7. Edit the program in the program editor, e.g. add a point.
8. Transfer changes to the controller:
   - Either transfer all changes:

     CONNECTION ribbon > **Upload programs to controller**
   - Or transfer only certain changes:

     Select one or more programs in the program editor > right-click > **Transfer changes**
   - Or transfer only the active program via the program editor:

   An overview opens: It shows the differences between the Sim project and OfficeLite.



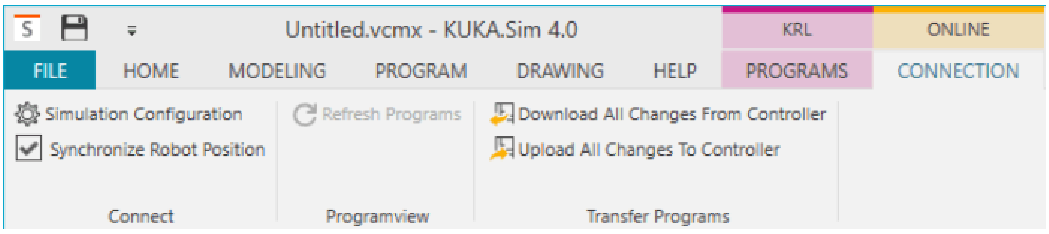
   Overview of "**Transfer changes**"
9. Conflict cases are highlighted and commented.

   For each conflict, decide how it is to be resolved: Click on the corresponding button.
10. Confirm the settings in the overview with **OK**.

    The data are transferred to OfficeLite. In the overview, a green check mark appears on the right-hand side of the lines that have already been processed.

    The overview closes automatically when the transfer has been completed.

CONNECTION ribbon



CONNECTION ribbon

Button	Description
Simulation configuration	Opens the **Simulation configuration** window.
Synchronize robot position	If it is not desirable for the changes to the robot position to be transferred from OfficeLite to KUKA.Sim: Uncheck the box.
Refresh programs	Only relevant for the mode **View only**: KUKA.Sim does not automatically detect when programs have been modified in OfficeLite. This button can be used to refresh the programs in KUKA.Sim.
Download programs from controller	Enables the program version of a robot controller (real or OfficeLite) to be downloaded to KUKA.Sim.
Upload programs to controller	Enables the program version to be transferred from KUKA.Sim to a robot controller (real or OfficeLite).

Simulating stopping distances

Simulating path-maintaining and path-oriented stops of the robot

The **Stopping distance configuration** window is displayed by default in the PROGRAM area. It can also be included in the view of other areas.

Path-maintaining and path-oriented stops of the robot can be simulated in KUKA.Sim. The 3D scene shows how and where the robot stops:

- Deviations from the planned path are shown as lines.
- Spatial deviations of the robot axes as well as the components are displayed as convex envelopes.

The area where the robot will move and where it will stop is represented as spheres with varying radii. The radius depends on the predictability: For example, the faster the robot moves, the larger the sphere, as the behavior cannot be predicted as reliably for a faster robot.

The safety tools and spheres defined in the safety configuration can be selected for determination of the stopping distances.

Precondition

- Running simulation (operator control elements active then only)
- Robot is selected in 3D scene.

Operator control area

Simulation configuration

- **Start position** / **End position**: A robot stop can be simulated for any position in the program or for defined start and end positions.

  If no positions are selected, the entire program is simulated.

- **Sampling rate**:

  The minimum sampling rate is 12 ms, as all calculations are based on the KUKA.RCS module.

  Default: 250 ms

Stop type selection

Selection of which stop type is to be simulated:

- **Only Stop 0**: EMERGENCY STOP
- **All non-path-maintaining stops**: Makes it possible to simulate path-maintaining stops with deviation paths.

Swept volume

Here it is possible to define which if any tools and geometries are to be tracked during the stops.

Additionally, the Sweep function can be used to visualize the position changes of the robot, the gripper/tools and the external kinematic systems.

Furthermore, different colors can be defined for the volumes.

Parameters for saving

The calculation results can be saved by assigning the generated geometry either to a new component or to the node of an existing component.

Visualization parameters

- **Deviation path**

  Deviation path can only be used in the **All non-path-maintaining stops** mode.

  The color profile for the display of the deviation paths can be defined here.

  By default, “no deviation” or “zero” means that the line is displayed in green. As the deviation increases, the color changes via yellow to red.

- **Activate calculation**

  Here it is possible to filter the volumes to be calculated. For example, it is possible to toggle between calculating deviation paths and swept geometry in order to calculate just spheres.

Reset: Resets the default values.

Configuring and starting trace recording

Description

Trace recordings are an important diagnostic tool during start-up of the industrial robot and during troubleshooting. With the trace functionality, different variables can be recorded while simulation is running, e.g. positions and velocities of individual robot axes or external axes.

> ℹ The trace recording can adversely affect the performance of the simulation. Depending on the computing power available, the simulation may not run smoothly with a large number of robots and external axes in the 3D scene.

Procedure

1. Activate the check box **Collect data** in the PROGRAM > **Trace** area.

   As long as this check box is activated, the data are recorded during the simulation.

2. Click on **Trace Diagram**.

   The window in which the 2D diagram of the recorded data is displayed graphically is opened.

   (>>> Trace Diagram)

3. Click on **Trace Settings**.

   The window in which the trace recording can be configured is opened.

4. Enter the desired parameters.

   (>>> Trace Settings)

5. Start the simulation.

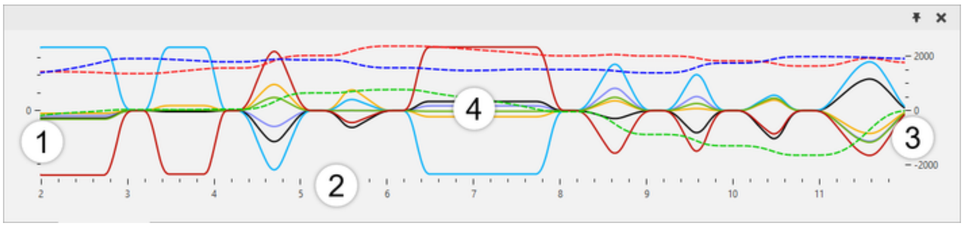
   The recorded data are displayed graphically in the **Trace Diagram** window.

Trace Settings

The settings are saved together with the project when it is saved.

Parameter	Description
Robot	Selection of the robot whose data are to be recorded. The desired robot can also be selected in the 3D scene and thus selected for the trace recording.
Show 3D line	If activated, the path of the robot is displayed in the 3D scene in the selected color.
Synchronise 3D time window	If activated, the path of the robot in the 3D scene is only displayed in the period selected in the 2D diagram.
Channel group 1	Group of data relating to the Y axis on the left-hand side of the 2D diagram.
Parameter	Selection of the data to be displayed in the 2D diagram: • **Axis position**: Indicates the positions of the selected axes. • **Axis velocity**: Indicates the velocity of the selected axes. • **Axis acceleration**: Indicates the acceleration of the selected axes. • **TCP position**: Indicates the position of the TCP in the WORLD coordinate system. • **TCP velocity**: Indicates the velocity of the TCP. • **TCP acceleration**: Indicates the acceleration of the TCP. • **BBRA**: Indicates the current override ($OV_ACT).
Upper limit	If activated, values that exceed the limit entered are shown in red in the 2D diagram and in the 3D scene. Values that do not exceed the limit are shown in gray.
Lower limit	If activated, values that fall below the limit entered are shown in red in the 2D diagram and in the 3D scene. Values that do not fall below the limit are displayed in gray.
Channel	Selection of the values (e.g. individual axes or external axes) to be displayed.
Color	Selection of the color in which the line of the corresponding value is to be displayed.
Weight	Selection of the thickness in which the line of the corresponding value is to be displayed.
Channel group 2	Optional: Group of data relating to the Y axis on the right-hand side of the 2D diagram. Selecting **None** deactivates the group. The parameters **Upper limit** and **Lower limit** can only be displayed for the first group. All other parameters must be configured identically.

Trace Diagram



Trace Diagram

Item	Description
1	Y axis for the data from: **Channel group 1** (solid lines) Click and hold down to shift the axis up or down. Scroll with the mouse wheel to zoom into or out of the display area of the axis.
2	X axis (time axis) As standard, the last 10 seconds of the simulation are displayed. Click and hold down to shift the axis left or right. Scroll with the mouse wheel to zoom into or out of the display area of the axis.
3	Y axis for the data from: **Channel group 2** (dotted lines) Click and hold down to shift the axis up or down. Scroll with the mouse wheel to zoom into or out of the display area of the axis.
4	2D diagram of the selected data. Click on a line to obtain further information about the exact point. The robot jumps to the selected time in the 3D scene. Scroll with the mouse wheel to zoom into or out of the 2D diagram. Click with the middle mouse button and hold it down to move the 2D diagram. Right-click > **Fit to view** to display all data at a glance. Right-click > **Center Y-Axes** to center the Y axes. Right-click > **Reset diagram** to restore the standard view of the 2D diagram.

Changing the tool during or after the simulation has no effect on the values of the trace recording. In order to perform the trace recording with a different tool, the tool must be changed and the simulation reset and restarted.

Intended use and misuse

Use

The KUKA.Sim software is used for creating 3D layouts with KUKA robots and also for the offline programming of KUKA robots.

Misuse

Any use or application deviating from the intended use is deemed to be misuse and is not allowed. It will result in the loss of warranty and liability claims. KUKA is not liable for any damage resulting from such misuse.