The JUNG Visu Pro System

Content:

1 OVERVIEW....2

2 NEW FEATURES VISU PRO....3

3 MAIN ASPECTS....4

3.1 Ease of Use and Comfort....4

4 TECHNICAL REQUIREMENTS....5

4.1 Hardware....5
4.2 Operation Systems....5
4.3 Internet Explorer, DirectX and .Net Framework....5

5 LOGICAL SYSTEM STRUCTURE....6

5.1 KNX and OPC Process Interfaces....8
5.2 Process Models....8
5.3 Calendar....8
5.4 Visualization....8
5.5 Control Panel....8

6 OPERATION MODES....9

6.1 Activated Product Version....9
6.2 Not Activated Product Version....9

7 COPY PROTECTION AND ACTIVATION....9

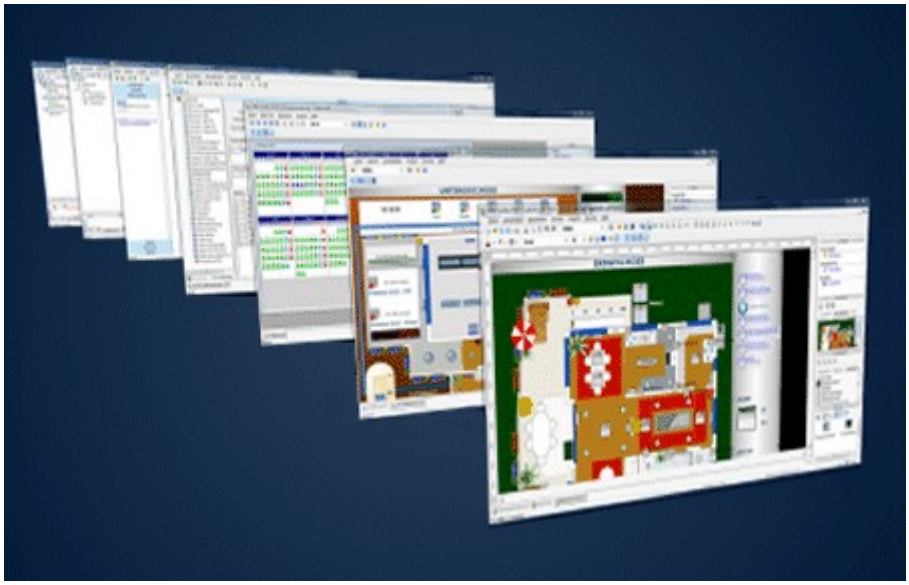
8 DISC STORAGE ORGANIZATION....10

9 BACKUPS AND EMERGENCY OPERATION....11

10 ADDRESS....12

1 Overview

The **visualization system** is a package of several WINDOWS applications with a consistent user interface for the visualization, control and surveillance of technical processes, in particular in the area of building automation.



With the help of process images, the system allows the user to overview and control technical processes, to archive data and to configure an alarm management system.

Moreover, it can be used as an automatic controller which automatically responds to events in the technical processes or influences the technical processes based on a time schedule. The time schedule can be defined with a calendar application based on day programs which are assigned to week days or special days, i.e. holidays, maintained and calculated by the calendar application.

The automatic behavior can be configured by means of "virtual devices" which are defined with the help of drag and drop. This is the easiest and preferred way to define automatic behavior. However, the system also comprises an integrated development environment for basic scripts, similar to Microsoft’s Visual Basic for Applications (VBA).

The visualization system comprises a direct connection to the KNX bus with a KNX editor which also helps putting KNX into operation and keeping KNX communication under surveillance. As an option, there is also an OPC client application available, which allows to connect to various other technical processes.

Other functions relate for instance to the user administration, with rights assigned to user groups, the secured logbook and the alarm management.

This whole set of functionality has been developed with ease of use and comfort as primary concerns.

2 New Features Visu Pro

Alongside various minor enhancements Visu Pro includes the following changes and bug fixes:

Visu Pro, JUNG Visu Pro Smart Assistant, project generating:

- The Visu Pro runner was added. With the starter it is possible to start projects at system startup.
- An update option has been added for generating JUNG Visu Pro Smart Assistant projects, so that updates of existing configurations / changes in the Visu Pro not be overwritten.

Visualization editor:

New:

- Element: **Timestamp**
  (see: Visualization editor help, 5.2.2.22)
- Element: **Numeric value input/output**
  (see: Visualization editor help, 5.2.2.20)
- Element: **Event/Interval chart**
  (see: Visualization editor help, 5.2.5)
- Element: **Color input**
  (see: Visualization editor help, 5.2.2.24)
- Element: **Operation mode switch over**
  (see: Visualization editor help, 5.2.2.12)

Changes:

- New **options** for the presentation of elements
  (Changable backgrounds, more layouts)
- The **visualization player** was **removed** and thus also the Active-X elements.
- The possibility of **basic script programming** was **removed**.

3 Main Aspects

3.1 Ease of Use and Comfort

The primary concern for the development of the visualization system has been ease of use for all phases of the project: planning, initial operation, regular operation, maintenance.

The system consists of different function modules with a clear structure which allows each module to focus on its individual task: configuration editors for process interfaces, editors for process logic, editor for process images, calendar program, system control.

All function modules share one unique philosophy for the user interface which, of course, meets modern WINDOWS standards. A user familiar with one function module will easily feel familiar with other function modules as well. This consequence in the user interface provides for the ease of use of the program.

Another tool to achieve ease of use is the application of context sensitive tools which always assist the user with information and functions according to the current operation.

Modern office programs have a quite simple subject compared to the control of technical processes which, in our opinion, should provide even better user interfaces in order to achieve the same ease of use. Therefore, we give first priority to the user interface.

4 Technical Requirements

4.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.
Interfaces	Interface for the connection to the process

4.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

4.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

5 Logical System Structure

The different applications of the **visualization system** package are organized in three levels:

Level of process interfaces:

The subject of this level is to configure and to observe the connection to technical processes in terms of devices and communication with devices. In order to set up the interface to a technical process, knowledge of these devices and their means of communication is required. The result is the ready configured process interface, which is maintained as a process interface project, i.e. a KNX or an OPC project.

Each process interface project forms a layer between the actual technical system with its devices at one hand and more application-oriented function modules at the other hand which move the focus from technical details of the actual devices toward logical relations and finally the presentation of certain aspects with process images.

Level of logical process models:

Whereas the process interfaces implement access to connected devices, the logical process model describes the desired functionality based on parameters given by these devices. At this level, additional parameters may be defined and automatic control functions may be set up to supplement the behavior of the devices.

For example, the process interface may provide access to a room's temperature and heating control, the logical process model will define archives of the temperatures, measured over a time period, and relate the desired room temperature to the occupancy state, perhaps planned in a calendar, by setting the control value for the heating device.

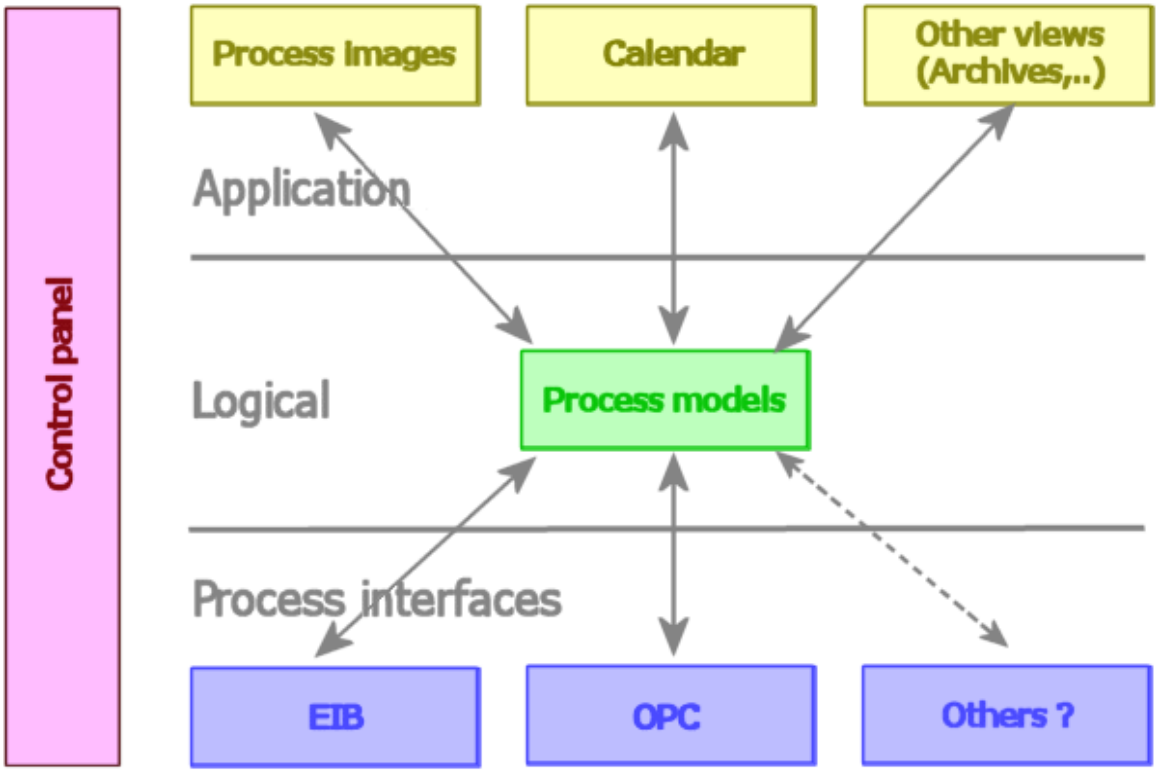
Several process interfaces may be combined in one logical process model.

This is also the level to define alarm and warning ranges for parameters and to set up the alarm behavior which may comprise sending notifications via email.

Application level:

The application level is based on the logical process model. It comprises views, for example process images, which provide a means to overview and control certain aspects of the system as well as the calendar program.

Access to these different levels and to certain functions is controlled by user rights. The user administration as well as other general settings for the visualization system, are defined in the control panel of the visualization system. The control panel provides access to all function modules of the system.


Control panel
Process Images
Calendar
Other views (Archives,..)
Application
Logical
Process models
Process interfaces
EIB
OPC
Others ?

5.1 KNX and OPC Process Interfaces

The KNX editor enables the connection of the system to the KNX bus. To this end, data can be imported from ETS to the KNX editor. Furthermore, the KNX editor provides functions for the operation and for logging.

Alternatively or additionally, the technical process may be connected via OPC (OLE for Process Control). For that purpose, a multitude of OPC servers is available which provide the process connection for the OPC client of the visualization system.

5.2 Process Models

In order to configure a process model, the process model editor imports data from the KNX or the OPC editor. A process model can integrate several different technical processes and enables the configuration of additional data points and logical functions.

The process model represents the logical aspect of the project, irrespective of the properties of the individual process interfaces. The process model allows to create "virtual devices", to define archives or to configure email notifications. Thus, it allows to configure the functioning of the technical system on the basis of the process interfaces.

For the execution of the project, an application for flow control is started which has no user interface. For this action the process model editor is not required.

5.3 Calendar

The calendar program allows the user to easily create and configure automatic time switch functions in connection with a calendar.

5.4 Visualization

The visualization via process images is implemented with the help of the visualization editor. The editor contains a multitude of functions for the configuration of the process images. It is not required for the execution of the project.

5.5 Control Panel

The control panel integrates the different function modules, documentations, the project management and settings for the system as a whole. It serves as a good starting point to access the different function modules.

6 Operation Modes

The software may run in one of two operation modes. The software must be activated for a certain computer via an activation code.

6.1 Activated Product Version

The visualization system is not limited by time or the number of program starts. It is only limited by the number of data points of the purchased product version.

6.2 Not Activated Product Version

The number of data points is given by the product version. After the software has been installed, it needs to be activated with an activation code. Until then, it will run for a number of days with a maximum of program starts.

Hence, the user may start using the software already before the software has been activated.

7 Copy Protection and Activation

The visualization system is copy protected. Copy protection is accomplished with either a dongle or an activation code.

In case of the activation code, the software calculates some characteristic data of the computer and shows up a system code. The software will be activated if an activation code is entered which matches the system code. Each system code and each activation code is unique and cannot be used more than once.

When characteristic data of the computer are changed, the system will ask for a new activation. However, main memory, network connection, graphics card or audio devices may be exchanged without impact on the activation state.

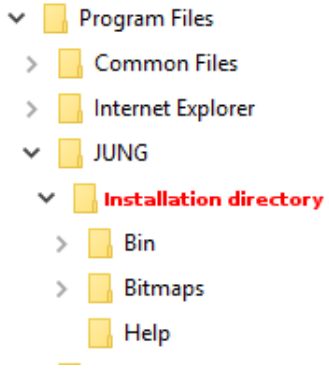
If the system asks for a new activation, the user will have several days to re-activate the software and in the meanwhile may continue using the software.

8 Disc Storage Organization

This description of the disc storage organization is only provided as supplementary information. Usually, knowledge of the directories used by the visualization system is not required to operate the system.

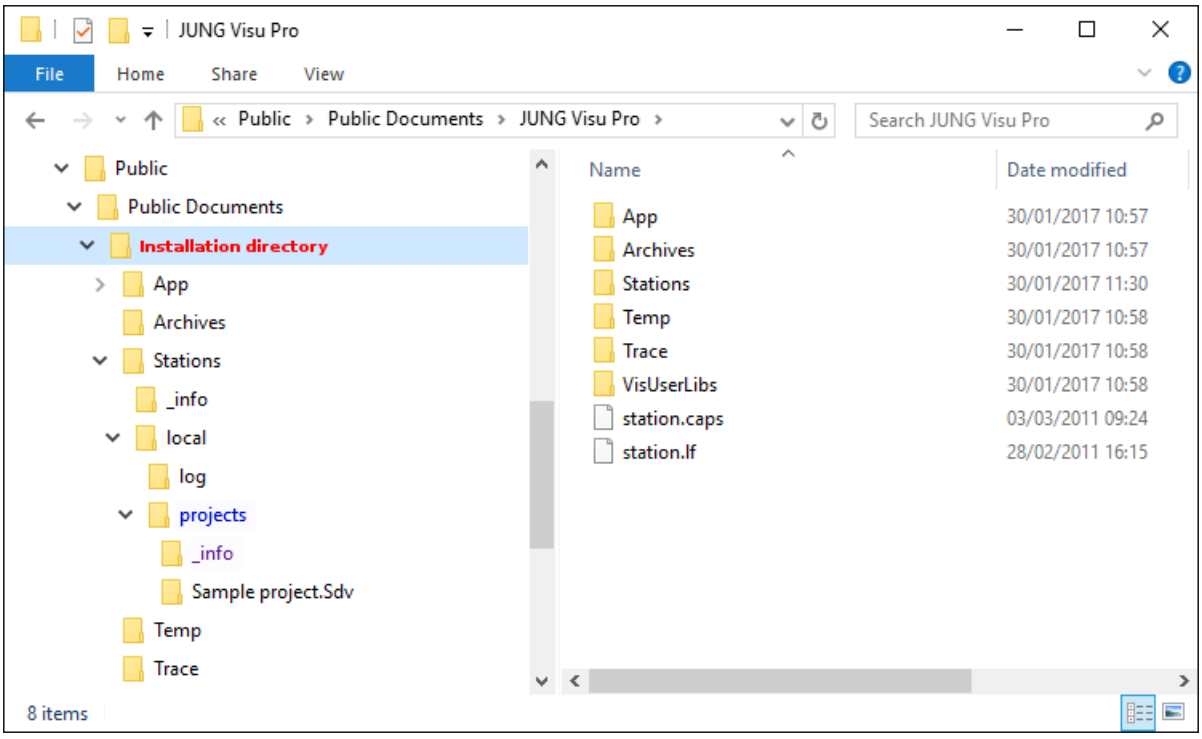
In particular in order to copy a project from one computer to another, the application of import/export functions of the control panel is recommended.

During installation, two directories are created: one for the software and one for all projects and other settings of the visualisation system.



The default installation directory for the **software** is created in the directory ***Program Files***. The help files and the graphics library are located in the corresponding subdirectories.

The directory **PROJECTS** is created in the default installation directory for the project data. You can change the directory during installation. Make sure, however, you have **write access** to the selected installation directory and the subdirectories!



Every project contained in the directory **PROJECTS** has a subdirectory which has the name of the project.

The subdirectory **_INFO** contains for every project a file with the project name.

9 Backups and Emergency Operation

In order to back up the data, export the projects periodically to a ZIP file using the export function in the control panel of the visualisation system.

In case your computer must be replaced, you can reinstall the software on a backup computer and then import the backed up projects using the import function in the control panel of the visualisation system.

Therefore it is useful to copy the graphics used in the project to the following directory:

***Installation directory*/Stations/local/*projects*/Projectname.VIS/images/**

Thus, you do not have to reconfigure the path specifications in the backup computer.

10 Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

JUNG Visu Pro
Control Panel

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....10

4 FUNCTIONS....11

4.1 Settings....11
4.1.1 User Management....11
4.2 Function Modules....14
4.3 Projects....15
4.3.1 Import Project....16
4.3.2 Export Project....19
4.3.3 Rename Project....20
4.3.4 Delete Project....21
4.3.5 Apply or Edit Project....22
4.3.6 Create Shortcut....22
4.4 Documents....23
4.5 System Log Files....23

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

The **control panel** provides an overview of the entire system, including its function modules, documents and the system log file. In the control panel you can define define project-specific settings as well as settings for the system as a whole.

The common Setting to all projects is the user administration.

Project-specific options comprise functions such as *delete*, *rename*, *export* or *import projects*. It also serves to open selected projects in order to edit their settings or to apply them.

The control panel is divided in the following categories:

- **Settings**: Here you can configure the user settings.
- **Function modules**: Contains an overview over the function modules of the system. You can start the modules from the control panel by double-clicking the respective module symbols.
- **Projects**: Contains an overview over all projects created in the system.
- **Documents**: Contains help files for all modules.
- **Log file**: Contains the entries of the system log file.

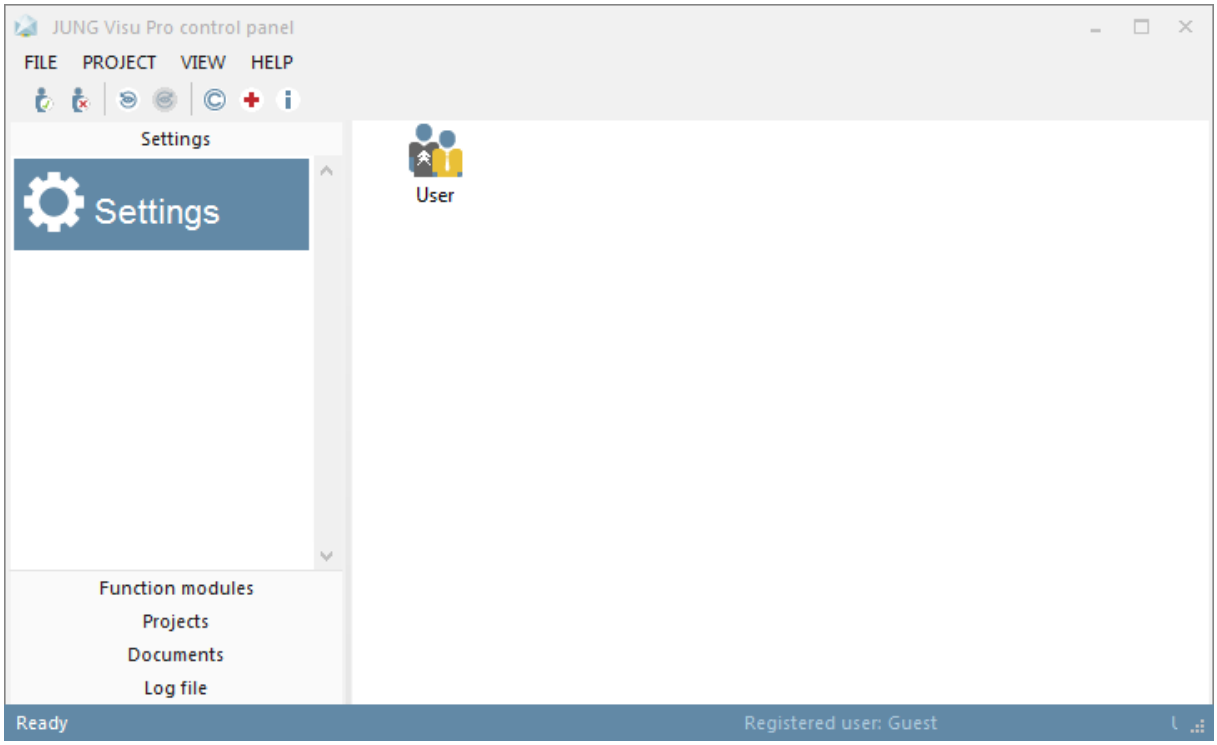
> ***Note:*** *The export and import functions can also be used to backup selected projects.*

After the installation of the **visualization system** a program group is created in the Start Menu. The default settings can be adopted or manually changed during installation. You may start the control panel application of the **visualization system** from the WINDOWS program manager. To start the individual modules, choose the tab ***Function modules*** and double-click the symbols of the respective modules.

First steps:

After the start of the control panel, choose the tab ***Settings*** to define general settings of the visualization system. The rights to change these settings are related to the user administration. If password protection has been enabled, the user must be entitled to perform these changes. When the system is started for the first time, password protection is disabled and the password for the predefined users is "System" (without quotation marks).

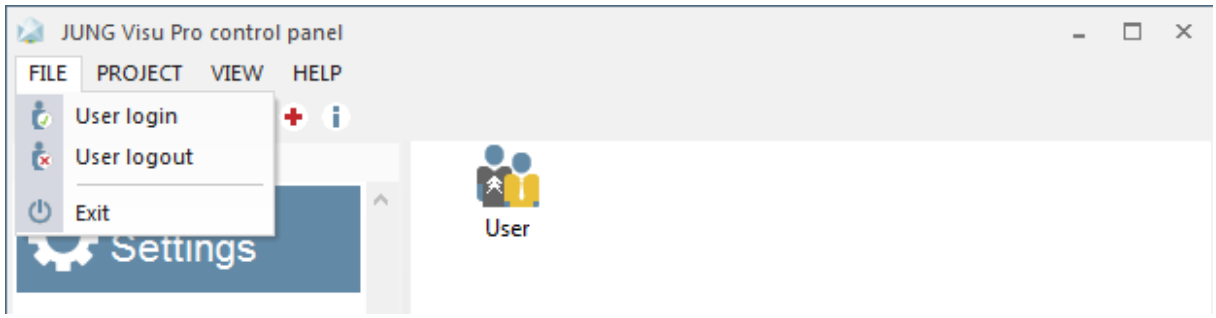
Of course, these passwords can be changed. Note that all passwords are stored internally as encrypted data and can hardly be retrieved in case of a loss. Therefore it is important, especially for the administrator password, not to forget passwords.



3 User Interface

3.1 Menu

Menu item *File*

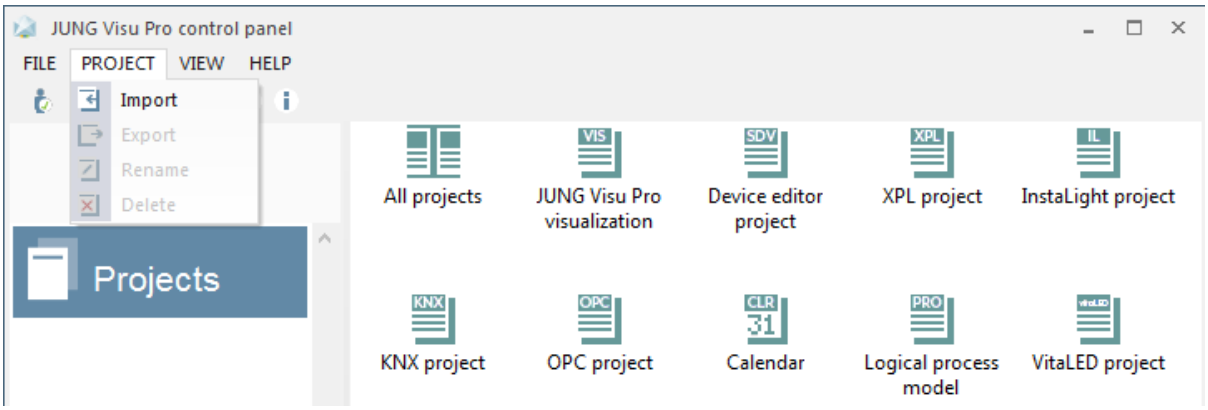


User login: Opens a dialog to log in a user to the system.

User logout: Logs out current user from the system.

Exit: Quits the control panel of the visualization system. Shortcut ALT+F4.

Menu item *Project*



Import: Opens a dialog to import a project from a zip archive.

Export: Opens a dialog to export the selected project.

Rename: Opens a dialog to rename the selected project.

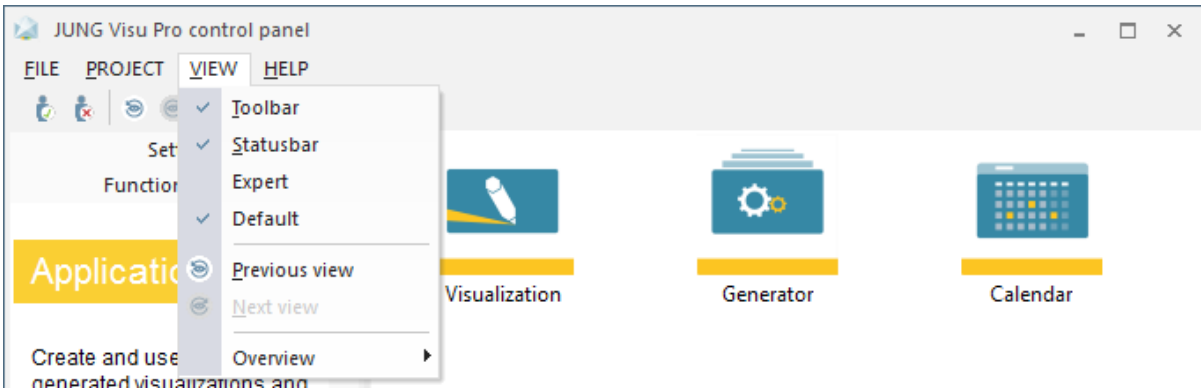
Delete: Opens a dialog to delete the selected project.

Apply: Applies the selected project.

Create Shortcut: Creates a shortcut on the desktop to open a visualization project or a calendar program.

Edit: Opens the selected project in its editor.

Menu item *View*



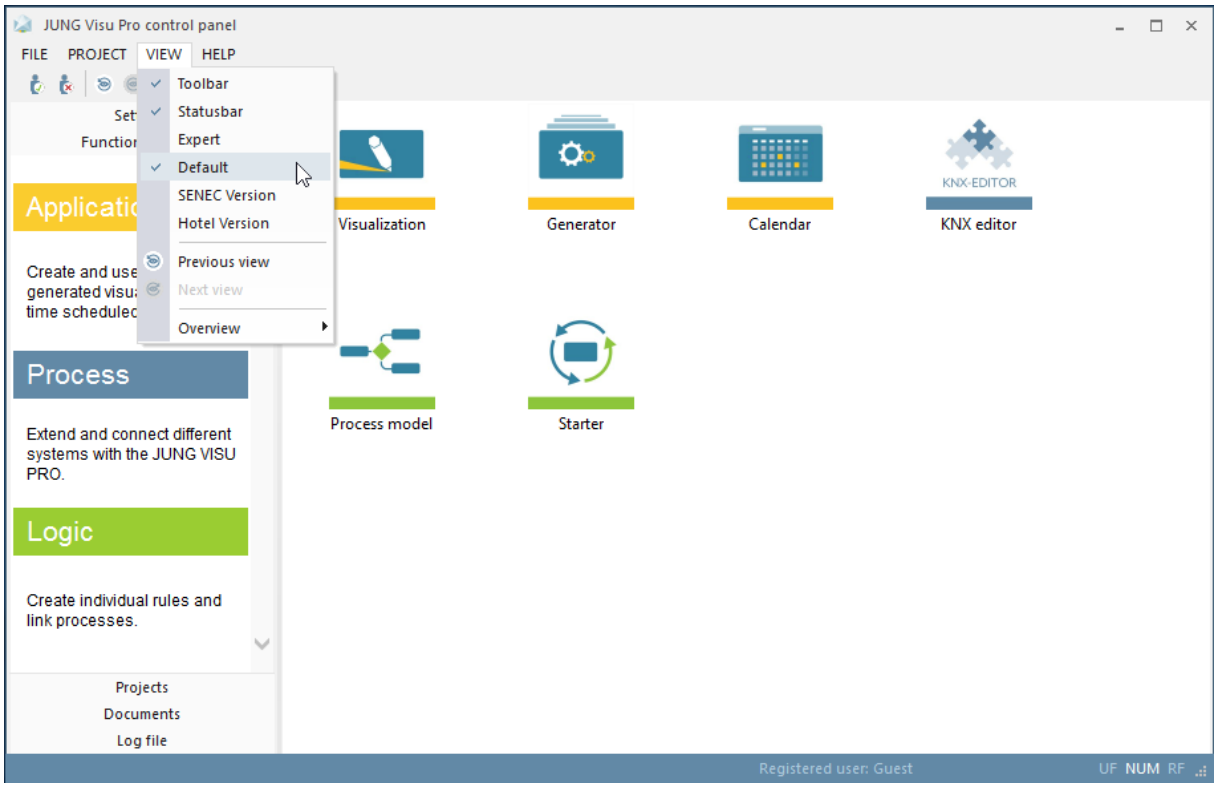
Toolbar: Toggles the visibility of the toolbar.

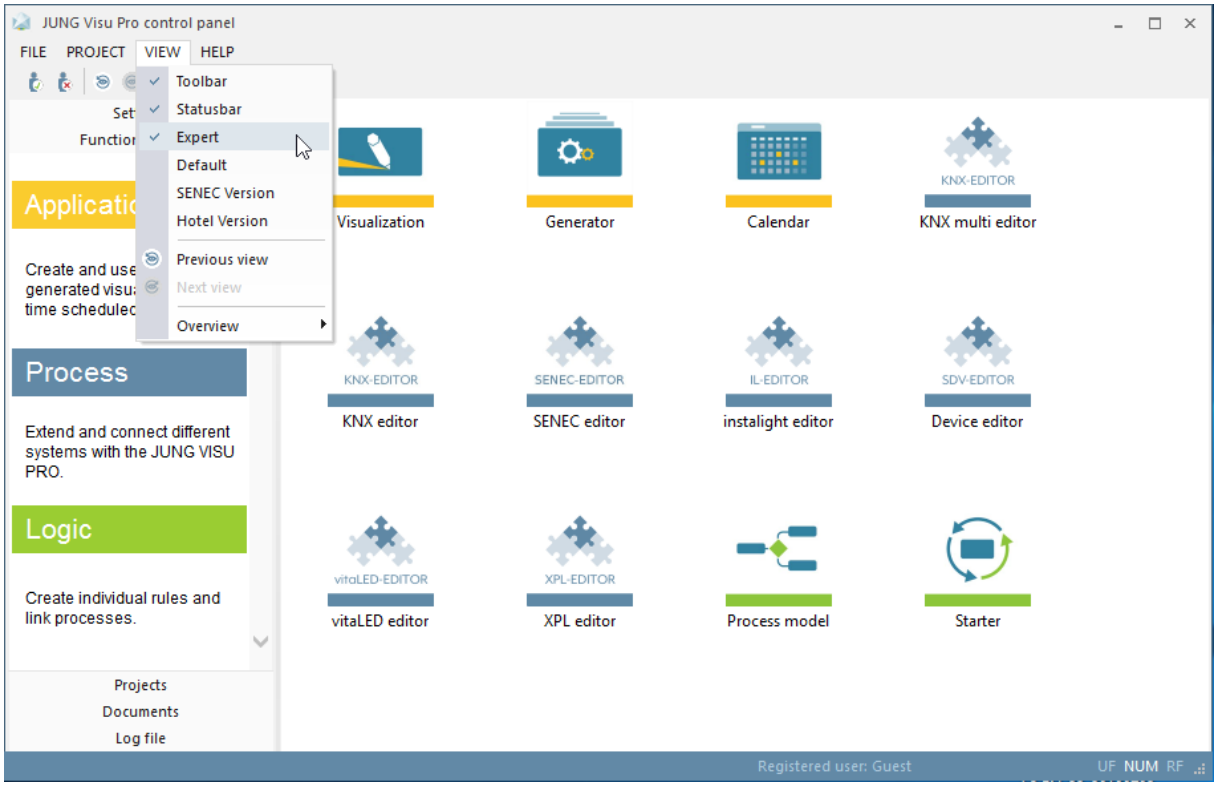
Statusbar: Toggles the visibility of the status bar.

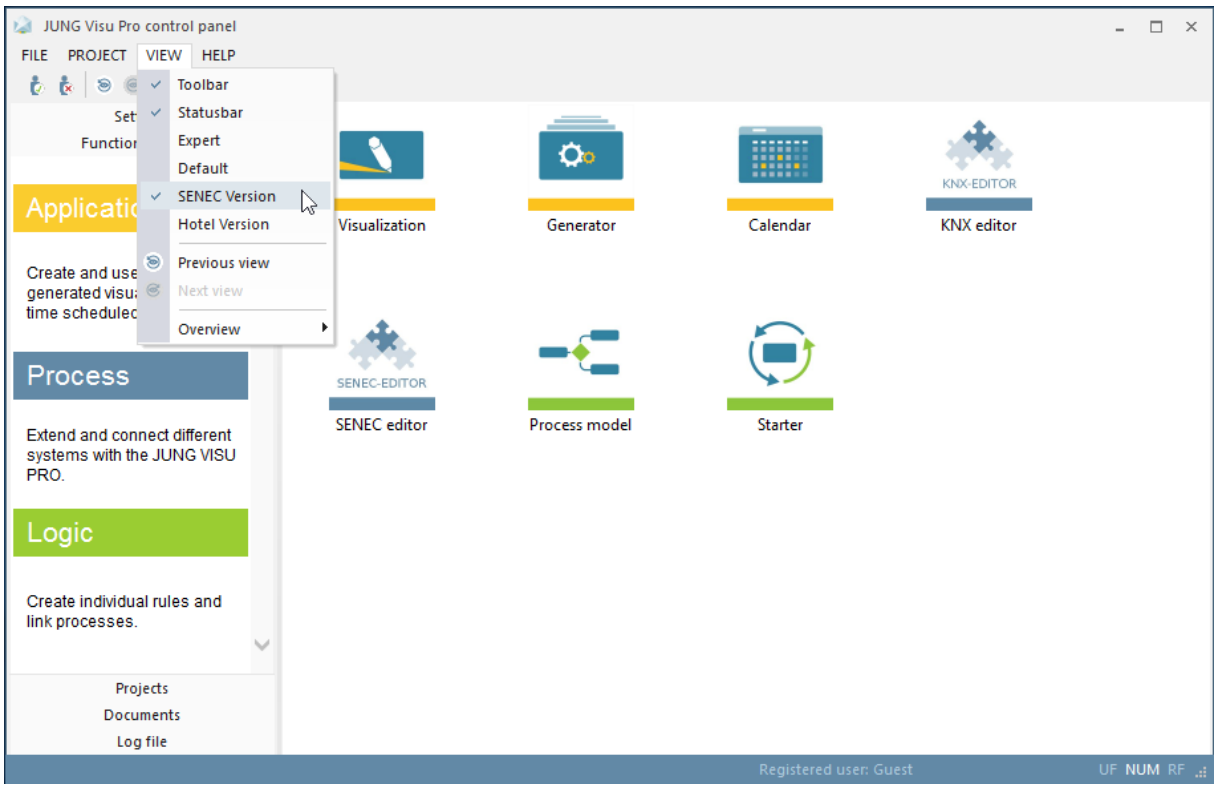
Default: Normally the *visualization editor, generator, calendar, KNX editor, process model editor* and *starter* are displayed.
In the SENEC version of the JUNG Visu Pro, the *SENEC editor* is displayed in addition to the standard function modules.
In the hotel version of the JUNG Visu Pro, the *KNX multi editor* is displayed in addition to the standard function modules.

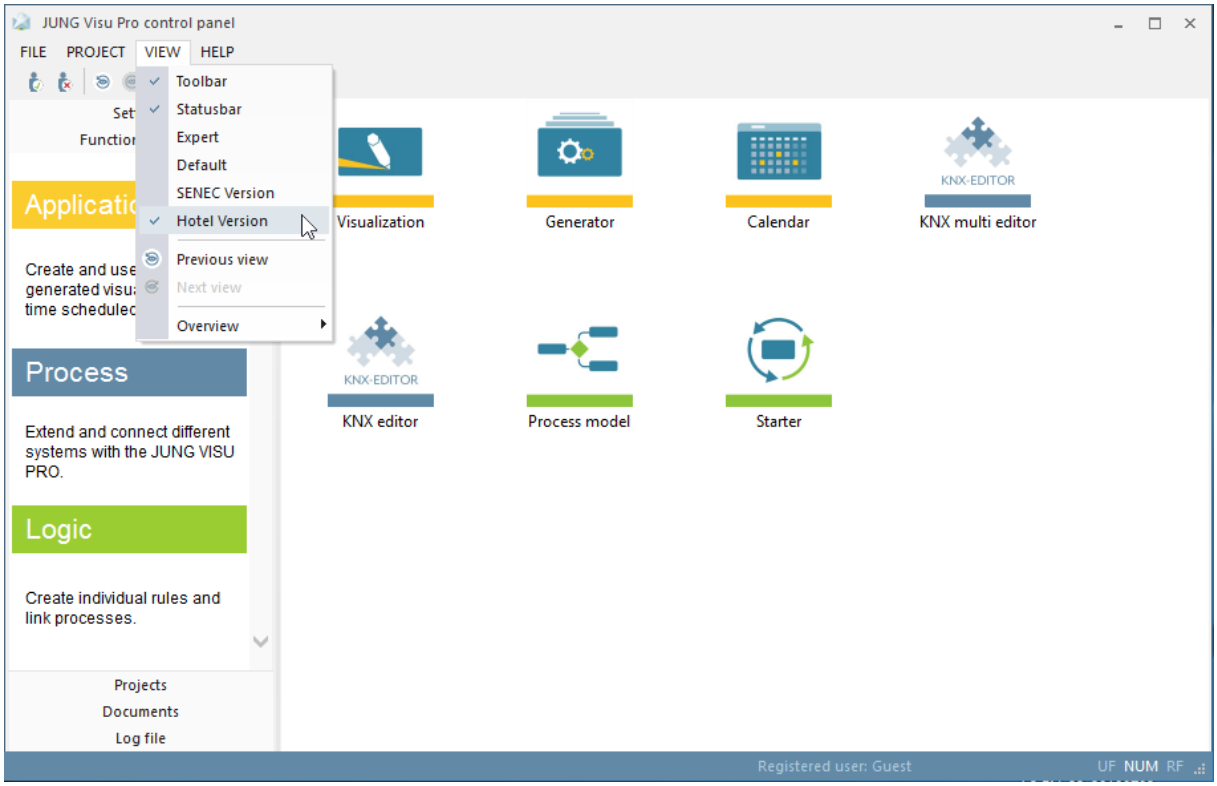
Expert mode: If the expert view is activated, the *instalight editor, device editor, vitaLED editor* and *XPL editor* are also displayed.

In the planner version of the JUNG Visu Pro you can choose between all views:




JUNG Visu Pro control panel
FILE
PROJECT
VIEW
HELP
Toolbar
Statusbar
Expert
Default
SENEC Version
Hotel Version
Previous view
Next view
Overview
Visualization
Generator
Calendar
KNX multi editor
KNX editor
SENEC editor
instalight editor
Device editor
vitaLED editor
XPL editor
Process model
Starter
Process
Extend and connect different systems with the JUNG VISU PRO.
Logic
Create individual rules and link processes.
Projects
Documents
Log file
Registered user: Guest
UF NUM RF


JUNG Visu Pro control panel
FILE
PROJECT
VIEW
HELP
Toolbar
Statusbar
Expert
Default
SENEC Version
Hotel Version
Previous view
Next view
Overview
Visualization
Generator
Calendar
KNX editor
SENEC editor
Process model
Starter
Process
Extend and connect different systems with the JUNG VISU PRO.
Logic
Create individual rules and link processes.
Projects
Documents
Log file
Registered user: Guest
UF NUM RF

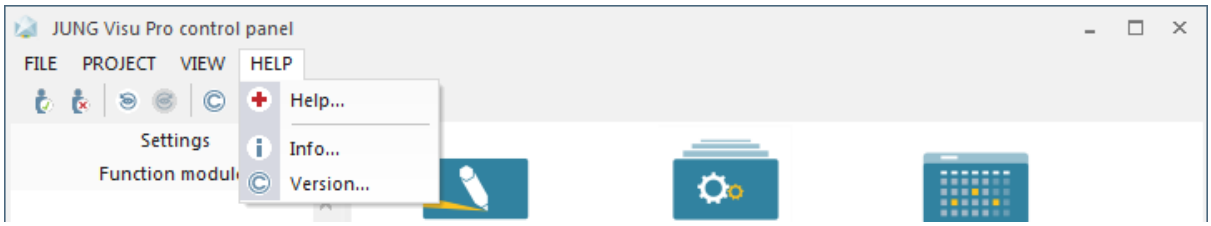

JUNG Visu Pro control panel
FILE
PROJECT
VIEW
HELP
Toolbar
Statusbar
Expert
Default
SENEC Version
Hotel Version
Previous view
Next view
Overview
Visualization
Generator
Calendar
KNX multi editor
Process
Extend and connect different systems with the JUNG VISU PRO.
Logic
Create individual rules and link processes.
KNX editor
Process model
Starter
Projects
Documents
Log file
Registered user: Guest
UF
NUM
RF

Previous view: Switches to the previous view.

Next view: Switches to the next view.

Overview: Switches to the view of a selected category.

Menu item *Help*

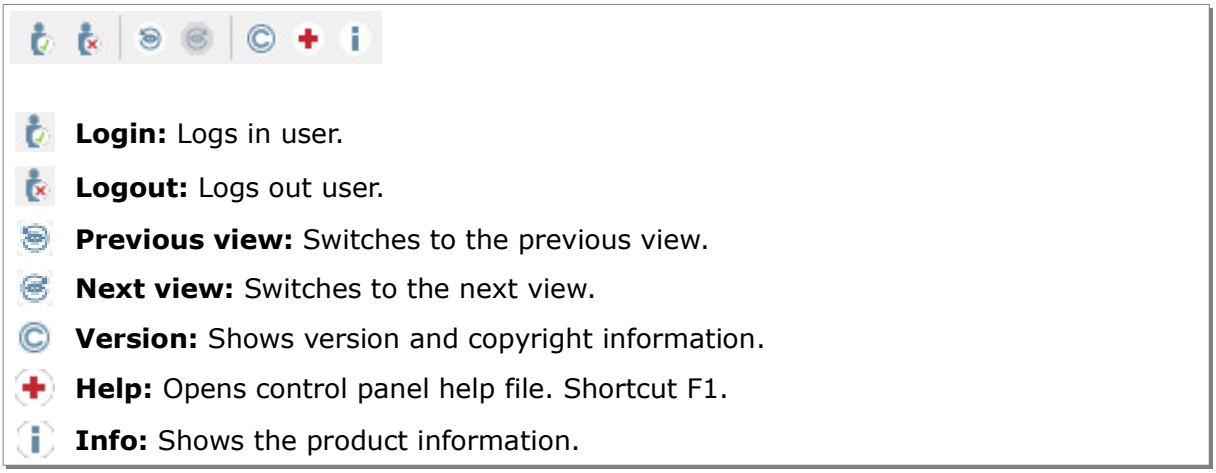


Help: Opens help file. Shortcut F1.

Info: Shows product and license information.

Version: Shows version and copyright information.

3.2 Toolbar



Login: Logs in user.

Logout: Logs out user.

Previous view: Switches to the previous view.

Next view: Switches to the next view.

Version: Shows version and copyright information.

Help: Opens control panel help file. Shortcut F1.

Info: Shows the product information.

4 Functions

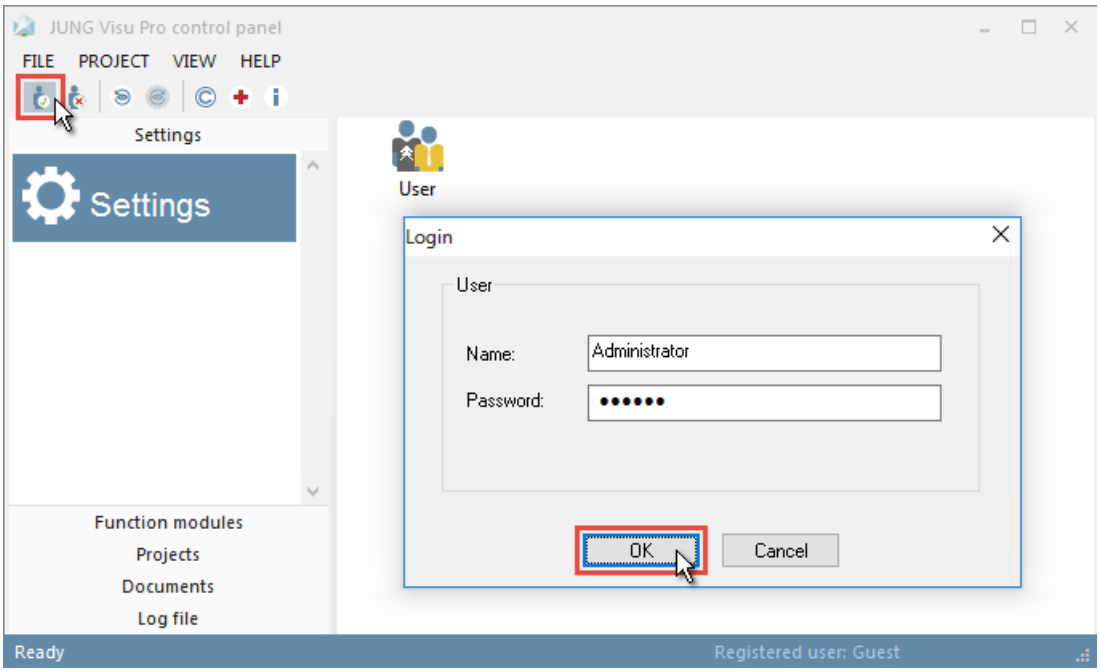
4.1 Settings

4.1.1 User Management

Double-click the symbol "User" to open the dialog ***User management***.

When password protection has been enabled, the user must be logged in with administrator or main user rights to access the user management.

To log in, choose the menu item ***File*** and click ***User login*** or click the button in the toolbar. The dialog ***Login*** opens.



In the login dialog enter the user name and the password, then click ***OK***.

Name: ***Administrator***
Password: ***System***

Enable/disable password protection:

To enable or disable password protection, select in the user management dialog the tab ***Settings*** and select or clear the checkbox ***Activate password***. We recommend to deactivate password protection until you are familiar with the features of the user management.

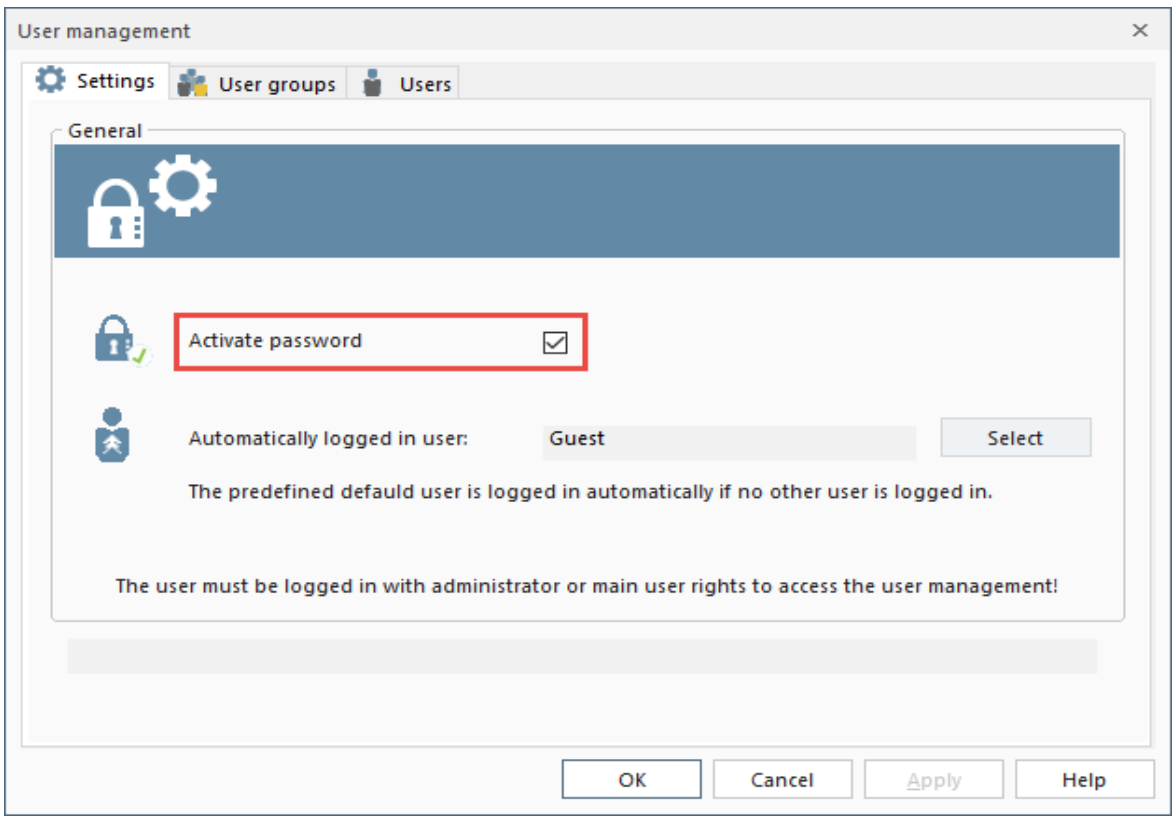
By default, password protection is turned off. However, the distinction into different user groups and users is effective even when password protection is disabled, in particular for the process visualization, since a starting behavior can be defined for a specific user group.

When password protection is turned on, a hierarchical system of user groups is activated, in which individual users may belong to one or more user groups.

The rights of the individual user groups are defined in the different function modules (process interfaces, logical process models, visualization, ..)

The highest level is the group of the system administrators, followed by the group of the main users, then the standard users, then the guests. In particular for the application with the process visualization, further groups may be defined and thus enable a finer control of user rights.

As a basic principle, system administrators always have the rights of main users, main users have all rights of standard users and standard users have all rights of guests.

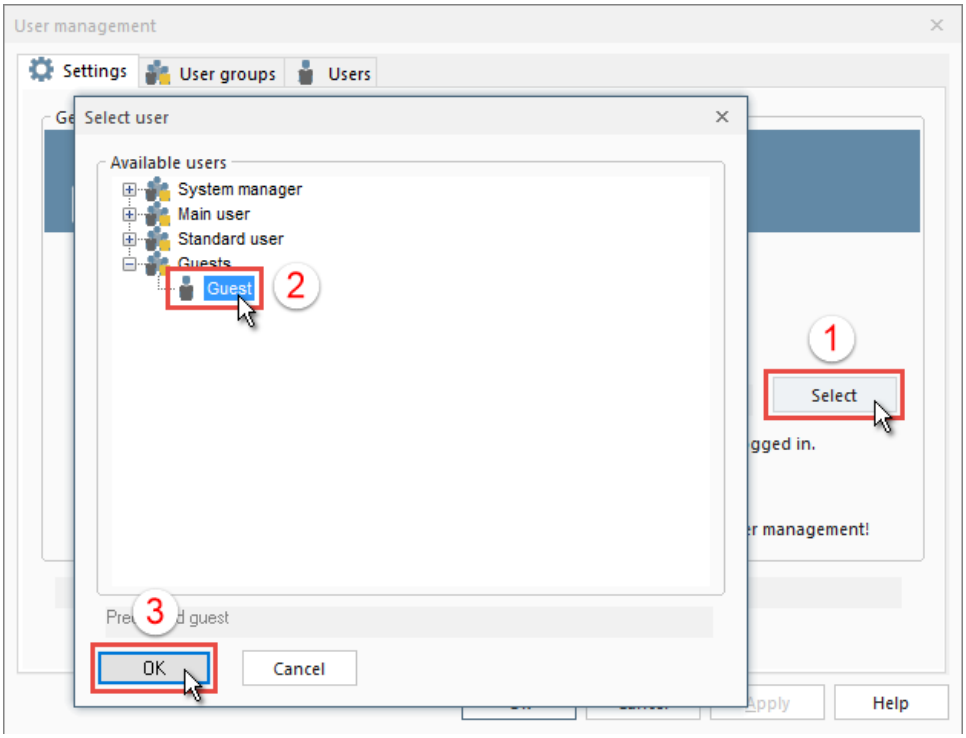


Select default user:

As a basic principle, there is always an active user, even if no user has been logged in explicitly. After the installation of the system, this is the predefined guest, but any user can be chosen to be logged in automatically instead.

When another user has logged in and later logs out, the predefined default user is logged in again automatically.

There is a relation between the automatically logged in user and the auto start feature. For example, when the computer starts, the visualization system can be configured to automatically start the visualization server with a specific worksheet which is the start worksheet for this user, if desired in full screen mode.

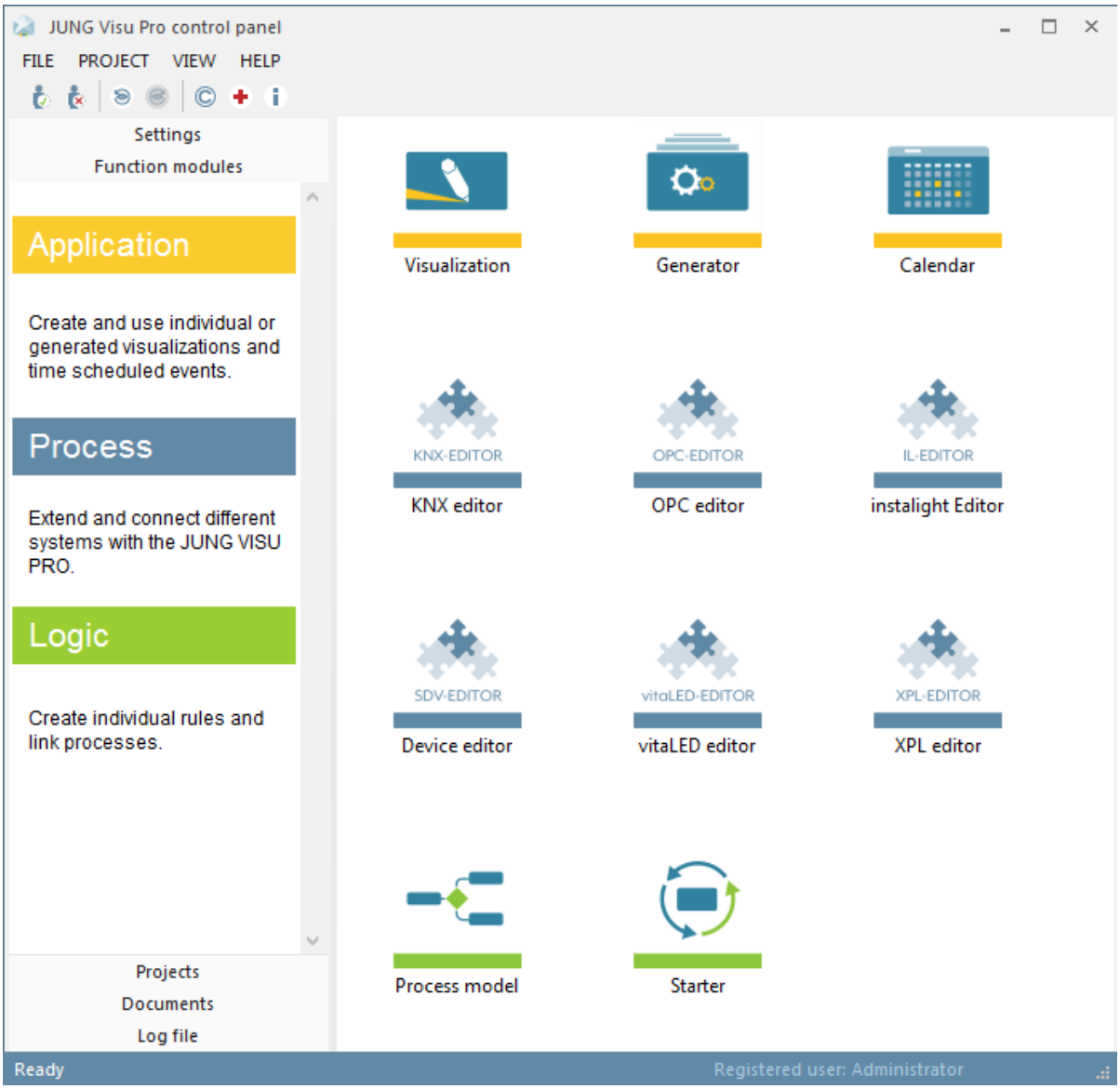


To change the automatic default user, select a user in the dialog and click ***OK***. Then the dialog will be closed. A shorter way is to simply double-click the user name.

In addition to the predefined users and user groups, more users and user groups can be defined after selecting the tabs ***User groups*** *or* ***Users****. Here the properties of existing users and user groups can be modified.*

4.2 Function Modules

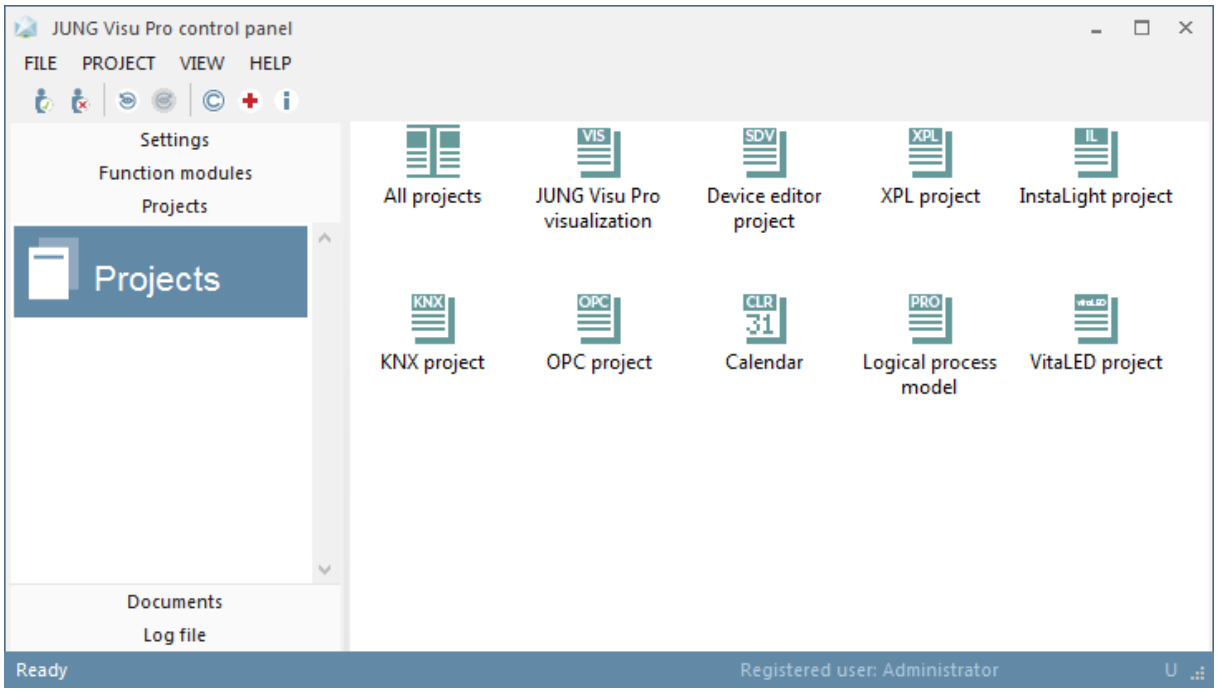
Select the category ***Function modules*** to see the different function modules of the system.
To start a function module, double-click its symbol.



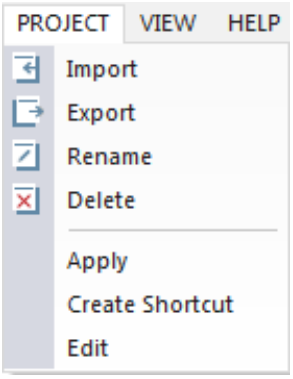
Note: If you can't see all function models, check whether the expert mode is disabled. (***View*** **>** ***Expert mode***)

4.3 Projects

To see the projects, first select the category ***Projects*** and then choose the project type by clicking the symbol of the project type. Click ***All projects*** to see all projects irrespective of the function module.



MENU ITEM PROJECT:



Import: Opens a dialog to import a project from a zip-archive.

Export: Opens a dialog to export the selected project into a zip-archive.

Rename: Opens a dialog to rename the selected project.

Delete: Opens a dialog to delete the selected project.

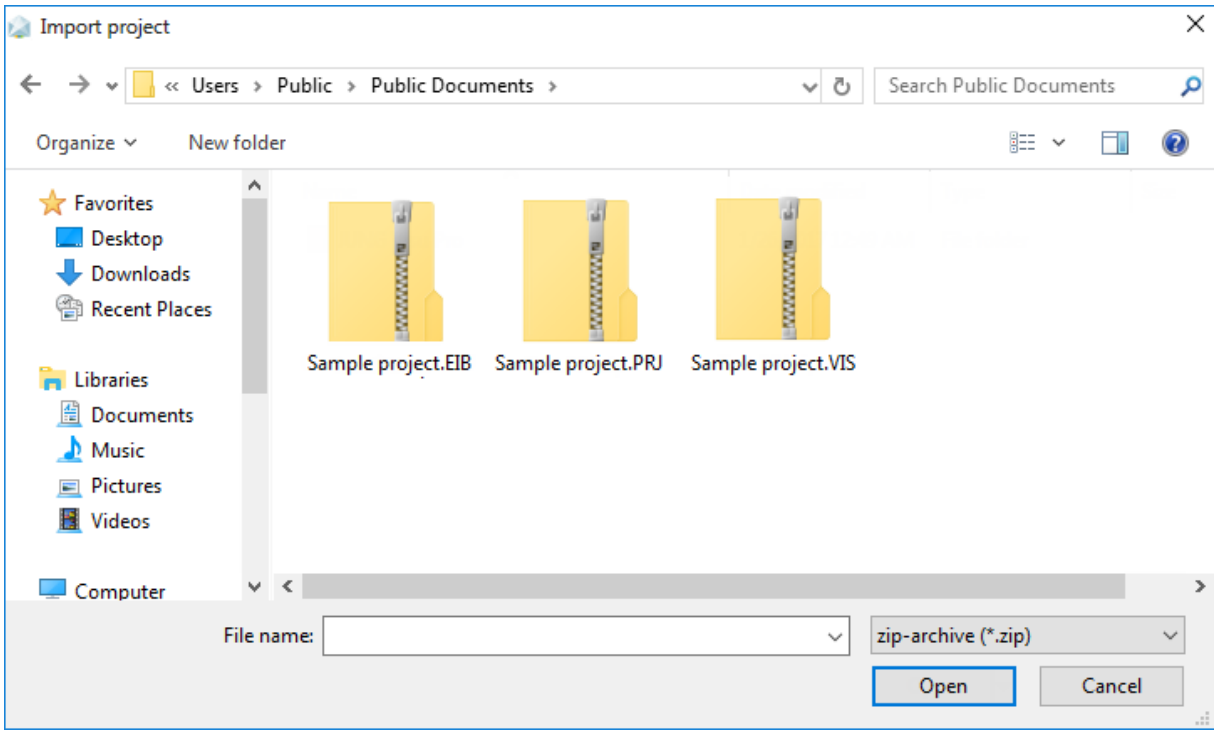
Apply: Applies this operation to the selected project.

Create Shortcut: Creates a shortcut of the visualization project or calendar program on the desktop.

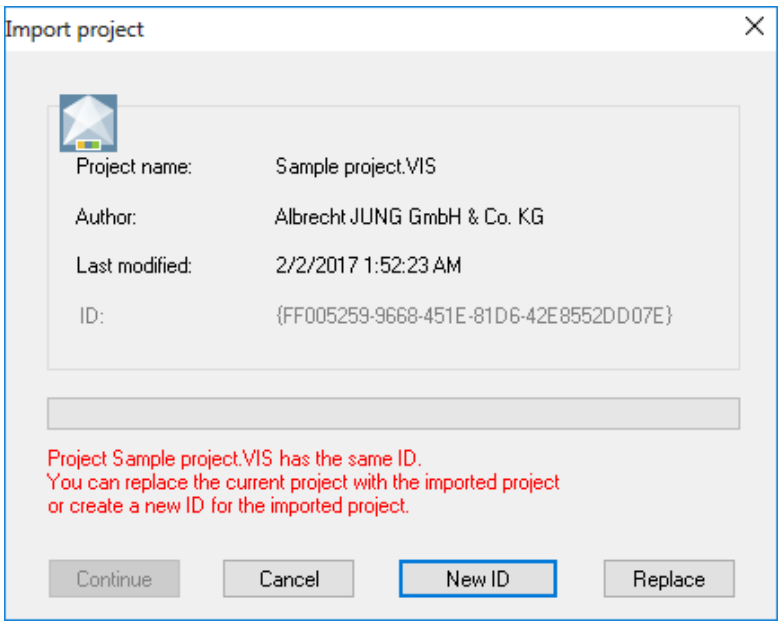
Edit: Applies this operation to the selected project.

4.3.1 Import Project

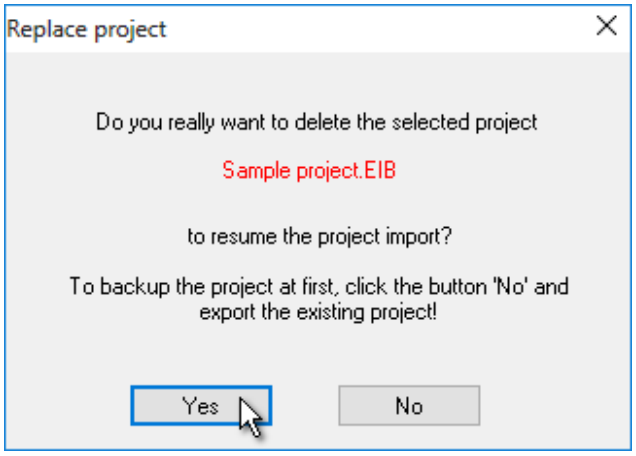
Choose the menu item ***Project*** and click ***Import***. The dialog ***Import project*** opens where a project can be imported from a zip-archive.



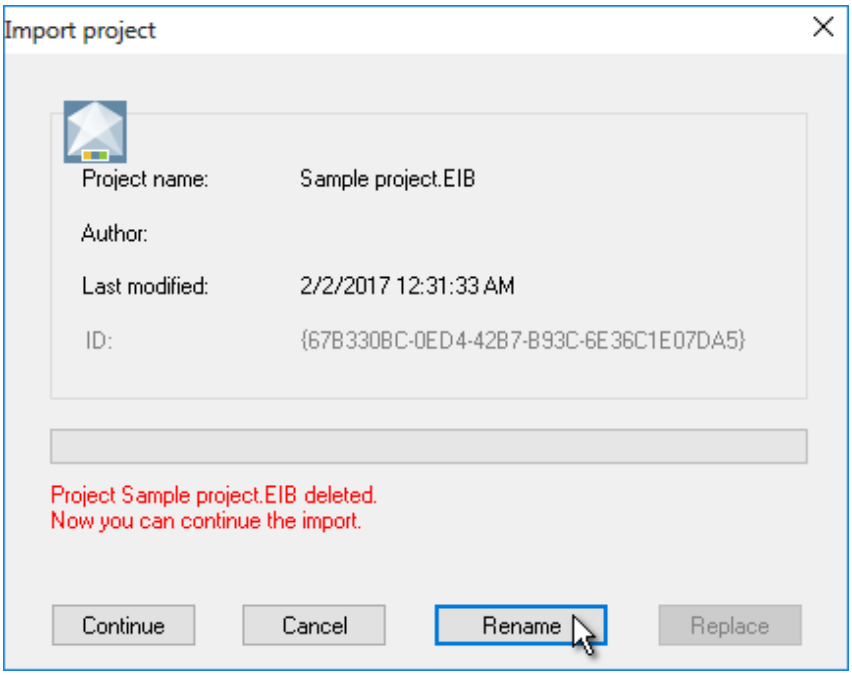
Select a zip-archive comprising a previously exported project and click the button ***Open***.



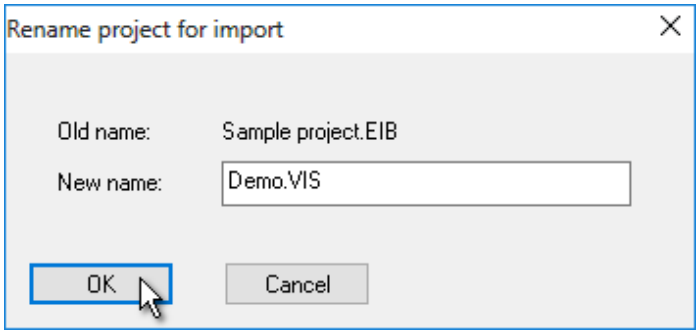
In this example a project with the same ID already exists. There is the possibility to replace the existing project.



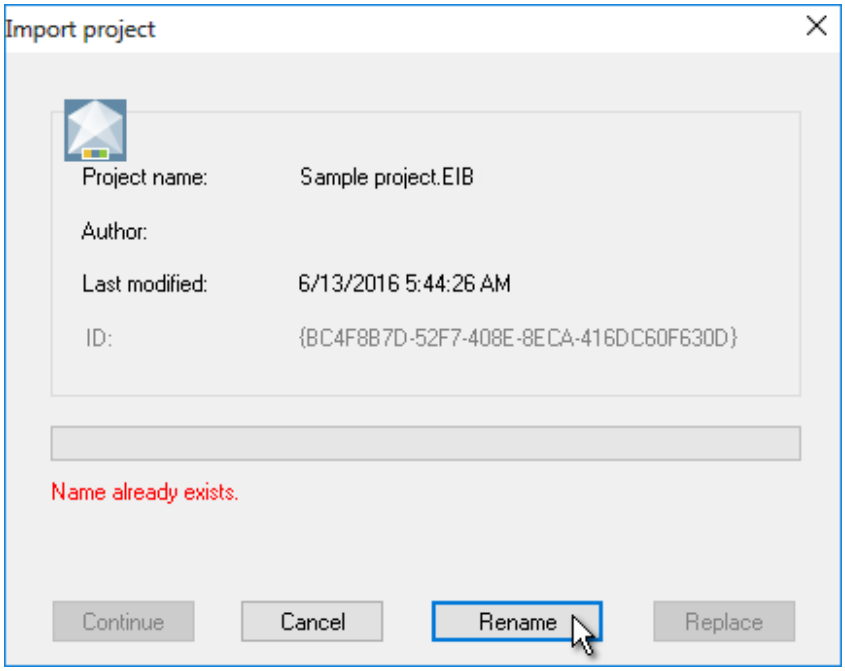
Click ***Yes*** to continue the import.

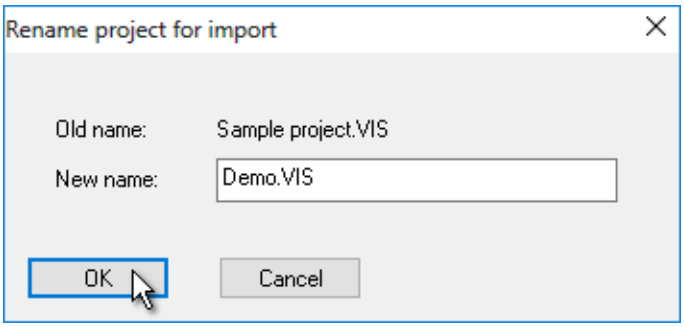


After the existing project has been deleted, click ***Continue*** to proceed with the import. Optionally, the project may be renamed before it will be imported. To rename the project, click the button ***Rename***.

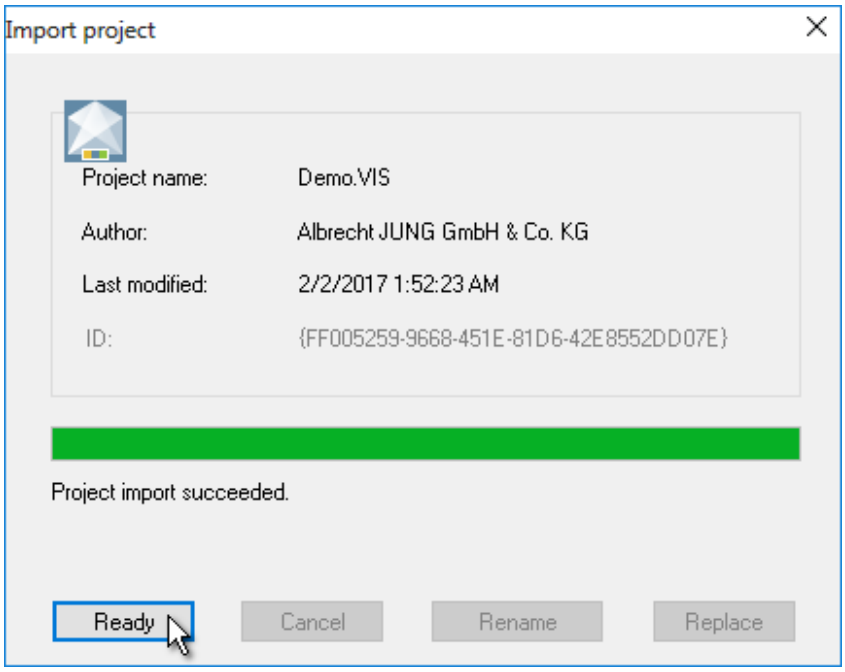


Enter the project name and click ***OK***. A project with this name may already exist, in this case choose another name.





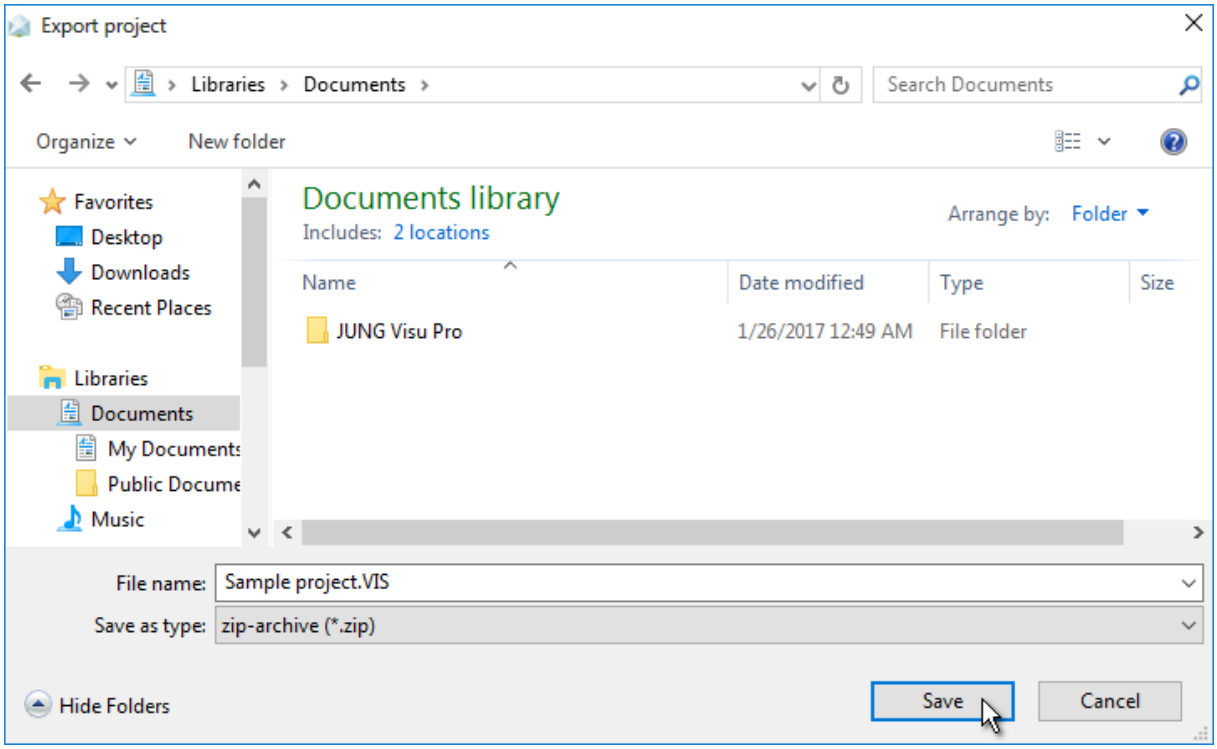
Enter another name and click ***OK***.



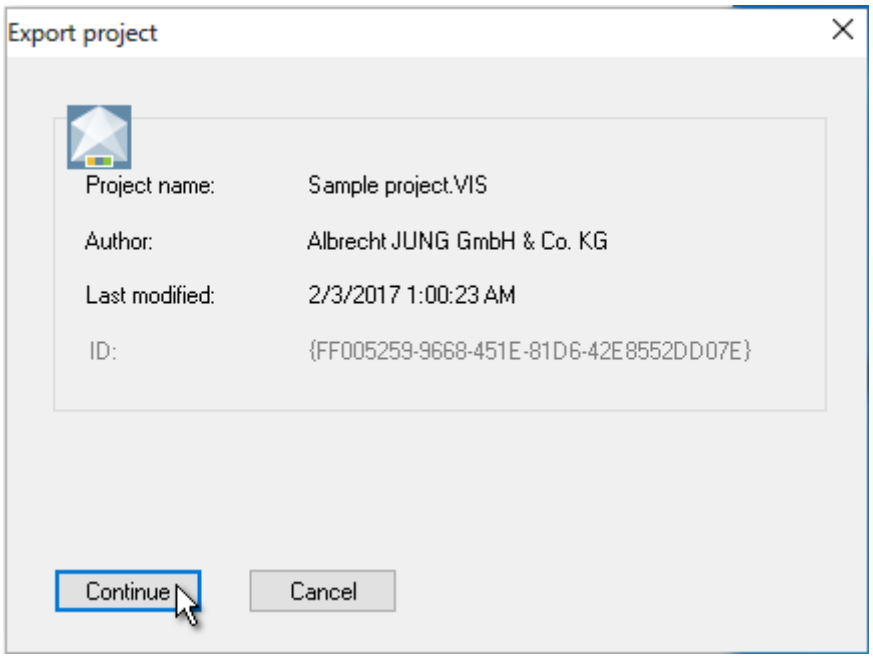
The project has been imported successfully. To acknowledge click ***Ready***.

4.3.2 Export Project

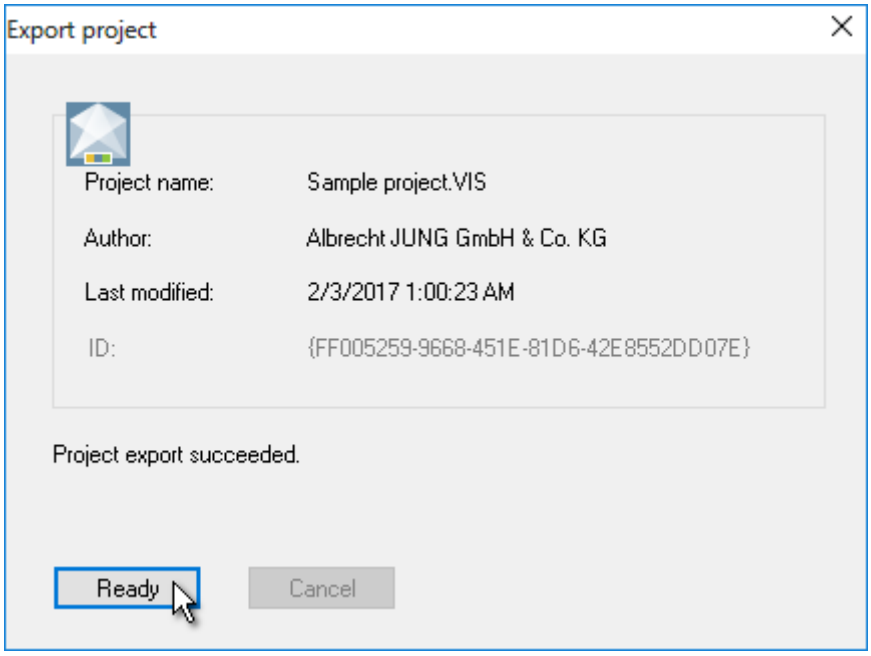
First select a project, then choose the menu item ***Project*** and click ***Export***. Alternatively, open the context menu with a right click on the project symbol. The dialog to export the project opens.



Select a directory and a name for a zip-archive, in which the project will be saved. Click the button ***Save*** to submit your selection.



Click the button ***Continue*** to proceed with the export.

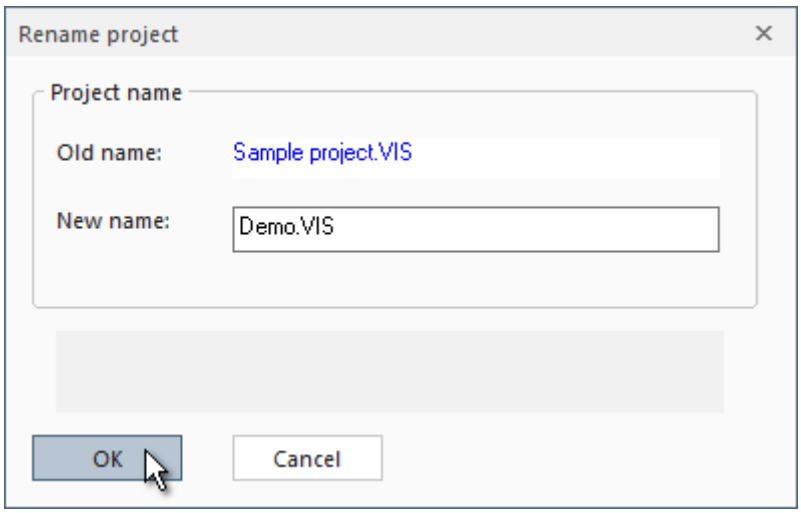


The project has been exported successfully.
Click the button ***Ready*** to acknowledge and to close the dialog.

4.3.3 Rename Project

Select a project, choose the menu item ***Project*** and click ***Rename***. Alternatively, open the context menu with a right click on the project symbol. The dialog ***Rename project*** opens.

> ***Note:*** *It is not possible to rename a project which is currently in use.*

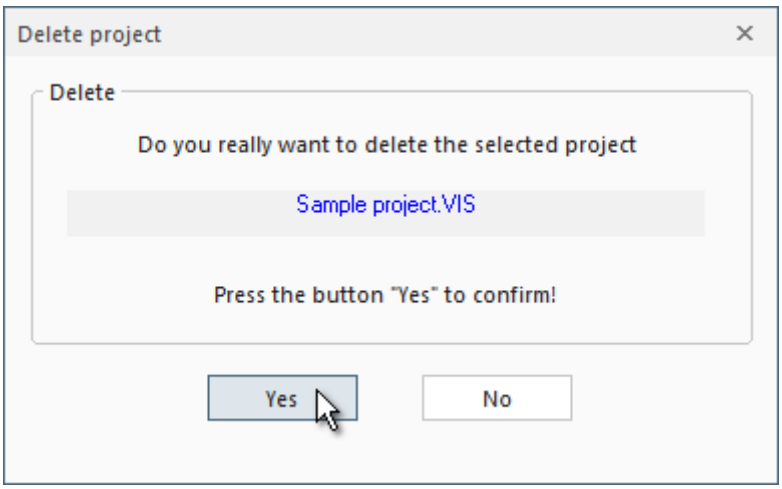


Enter the new name and click ***OK***.

4.3.4 Delete Project

Select a project, choose the menu item ***Project*** and click ***Delete***. Alternatively, open the context menu of the project with a right click on the project's symbol. The dialog ***Delete project*** opens.

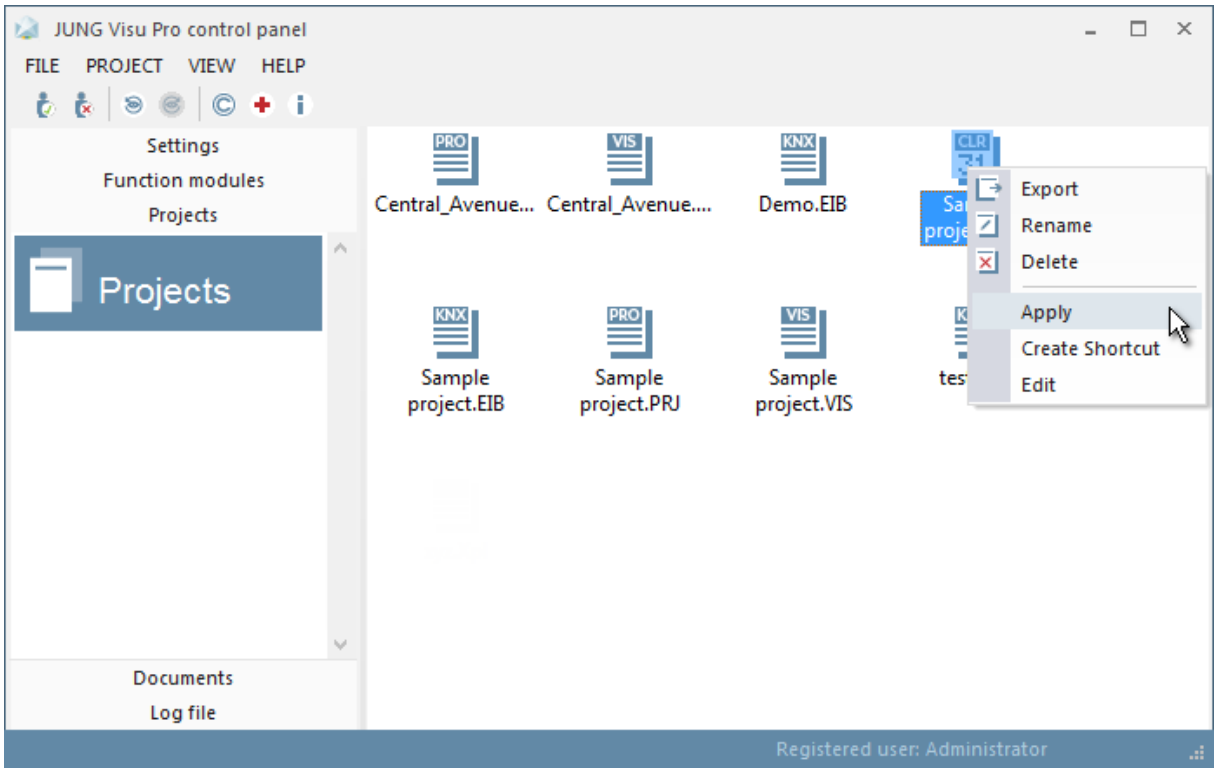
> ***Note:*** *It is not possible to delete a project which is currently in use.*



Click ***Yes*** to acknowledge the deletion of the project.

4.3.5 Apply or Edit Project

Choose the menu item ***Project*** and click ***Apply*** or ***Edit***. Alternatively, open the context menu of the project with a right click on the project symbol.



The context menu offers different actions which can be performed on the project. Just select the respective menu option to proceed with the desired action.

4.3.6 Create Shortcut

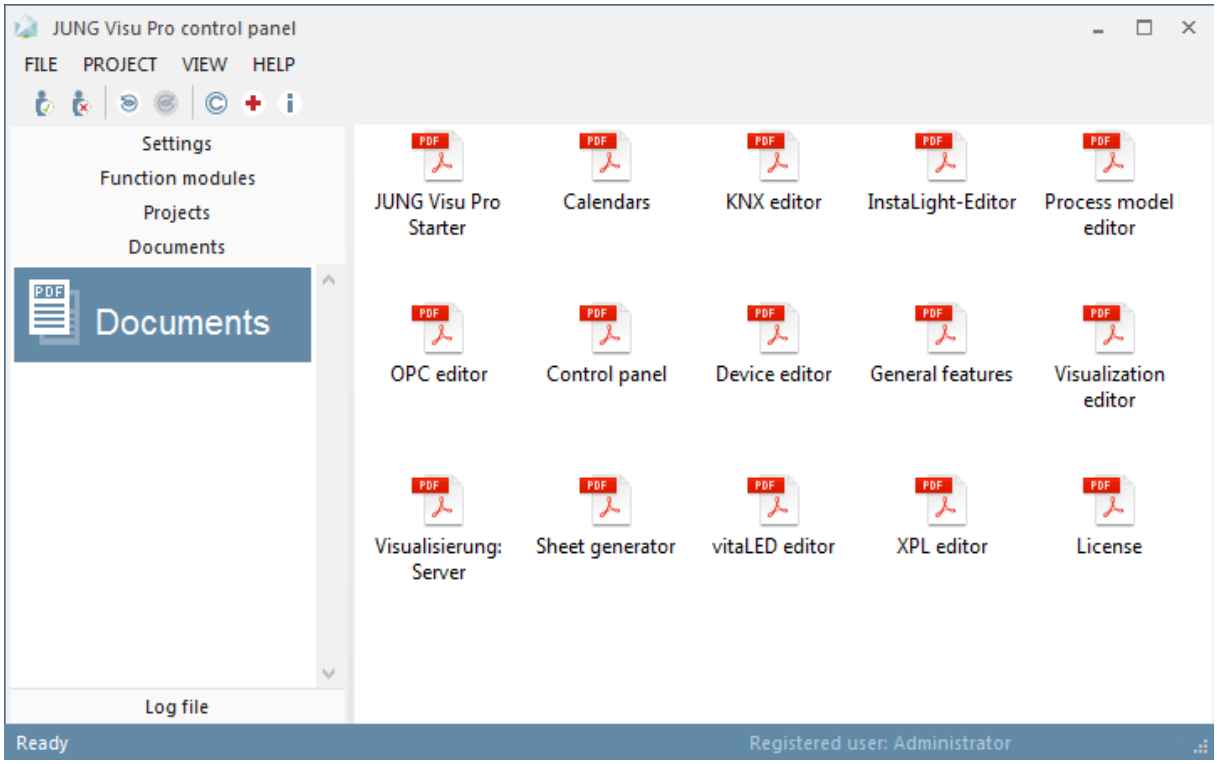
For calendar and device editor projects it is possible to create shortcuts on the desktop to directly open the project. To create a shortcut, choose the menu item ***Project*** and click ***Create Shortcut***. Alternatively, open the context menu of the project with a right click on the project symbol.

A double click on the shortcut of a visualization project starts the server with the corresponding project.

A double click on the shortcut of a calendar project opens the calender application with the corresponding project and starts the execution of the time program.

4.4 Documents

Choose the category ***Documents*** to get an overview of all help files and other documentation related to the visualization system. Double-click the file symbol to open the file. In order to open PDF files the Adobe Acrobat Reader® application is required. It can be installed from the installation CD of the visualization system.



4.5 System Log Files

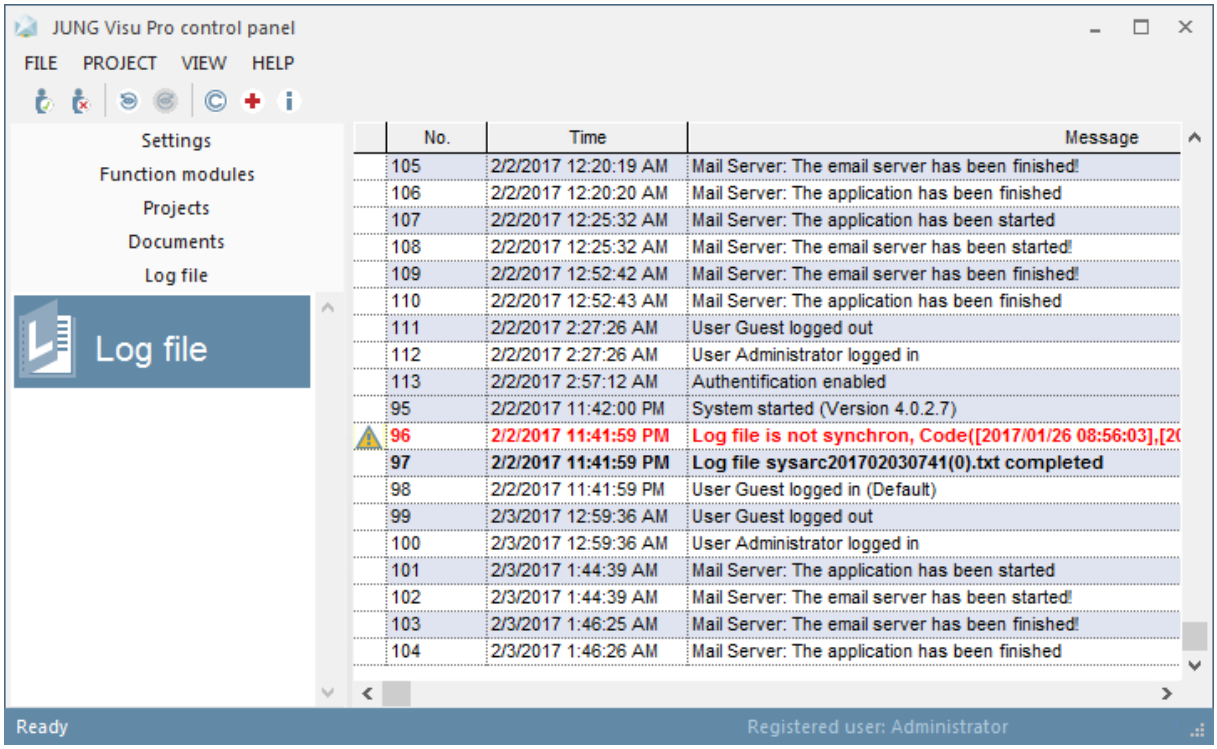
The control panel comprises a window to see parts of the system log file. The log file contains for example information about login/logout actions.

The log file can be important in the context of warranty.

Organization of system log files:

The system log files comprise a certain amount of files which are maintained as a first-in-first-out structure. Currently, the log files use up to 100 MB disk memory. The messages in each file are in text format and can be imported for instance into Microsoft Excel. Until the maximum of 100 MB disk space has been reached, more files will be created when the current log file has reached either a maximum size of about 1 MB or comprises 8000 messages at most. After the maximum of 100 MB disk space has been reached, older log files will be deleted to free disk space.

The control panel log file window shows the two most recent log files.



Security of system log files:

The individual log files can be easily examined because they are text files and can be loaded into a spreadsheet program or a text editor. The drawback could be that entries in the log files can be changed easily.

Still log files are secure, because all manipulations can be detected. The log files comprise encrypted check sums which protect the single messages and the sequence of messages.

In the control panel log file window manipulated parts of the log file are displayed as strikeout text.

If you overview the log file make sure that the following properties hold true:

a) *Numbers of messages are sequence numbers without gaps.*
b) *The messages are never displayed as strikeout text.*

Further measures can be taken in order to protect the log files. Ask us if you think that your log file has been tampered.

JUNG Visu Pro
Worksheet Generator

Content:

1 TECHNICAL REQUIREMENTS.....2

1.1 Hardware.....2
1.2 Operation Systems.....2

2 GENERAL INFORMATION.....2

2.1 First Steps.....3
2.1.1 Options.....4
2.2 Information about Generating.....8
2.3 Update Project Lists.....8

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ALBRECHT JUNG GMBH & CO. KG
All rights reserved.

Trademarks
Microsoft®, DirectX®, .NET Framework®, Windows®, Windows NT® are registered trademarks of the Microsoft Corporation.
All trade, company and product names are trademarks or registered trademarks of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

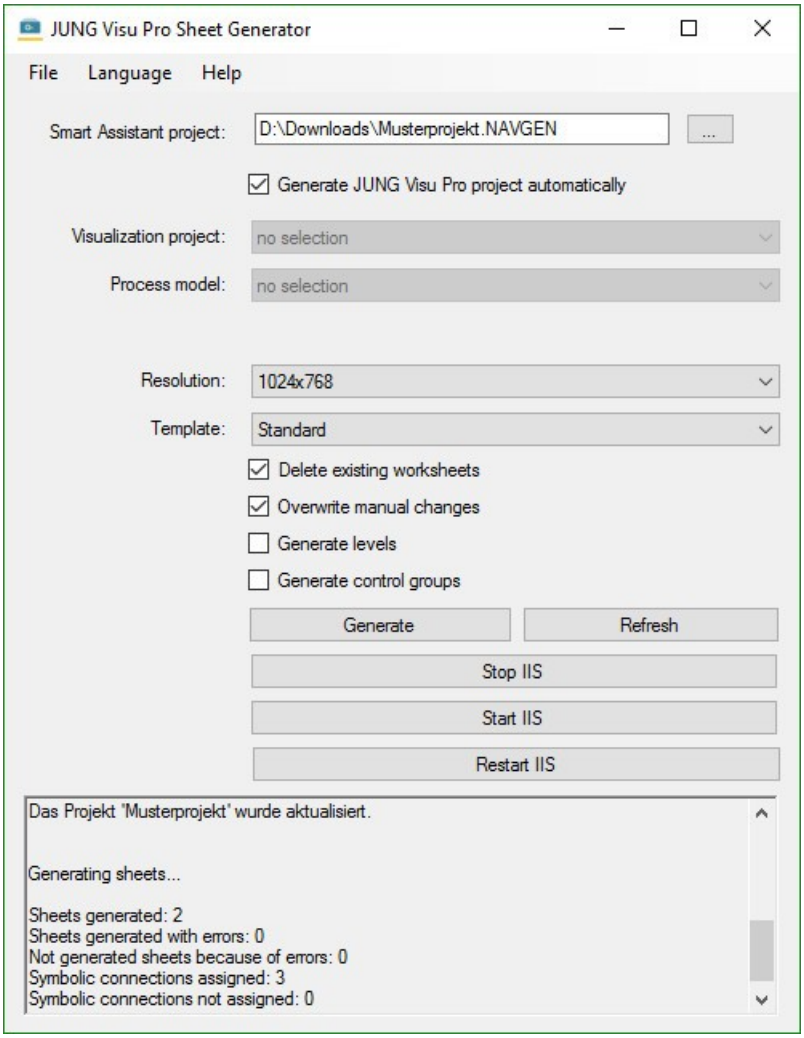
2 General Information

The JUNG Visu Pro Worksheet Generator is an addition to the JUNG Visu Pro Smart Assistant and is installed with the Visu Pro on the system.

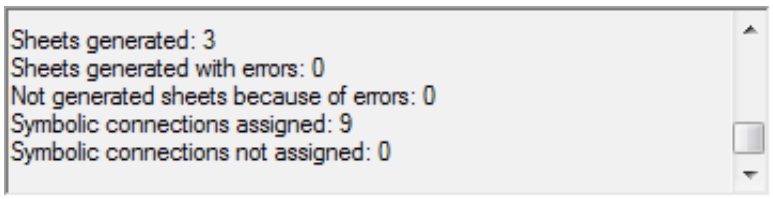
Projects which have been generated and exported via the JUNG Visu Pro Smart Assistant (***Projectname.NAVGEN***) can be converted to worksheets for the selected visualization project with the help of the worksheet generator.

2.1 First Steps

1. Open the generator.
2. Select a Smart Assistant project.
3. Choose the desired settings.
4. To create the worksheets, click the button ***Generate***.



The message list in the bottom part of the generator displays the performed actions.



When the visualization project is opened in the visualization editor, the generated worksheets will be available.

Note:

The template for the **Smart Control IP** is to find under the 440x800 resolution.
This template is compatible only with the Visu Pro from version 3.3 and the visualization server from version 2.0.

Recommended settings for smartphones: Resolution: 440x800
Template: Smartphone

Recommended settings for tablets:	Resolution:	1024x768
	Template:	Tablet 1 (levels/rooms right)
		Tablet 2 (levels/rooms left)

2.1.1 Options

The following options are selectable and influence the result of the generated project.

Smart Assistant project: Only files with the ending **.NAVGEN** are allowed.

Generate Visu Pro projects automatically: When activated, the KNX project and the process model will be automatically generated, or if already existing, updated (see also 2.2).

Visualization project: Select the existing project that should be updated. Is the check mark at 'Generate Visu Pro projects automatically' set, the option is deactivated because the generator uses the file name of the Smart Assistant project.

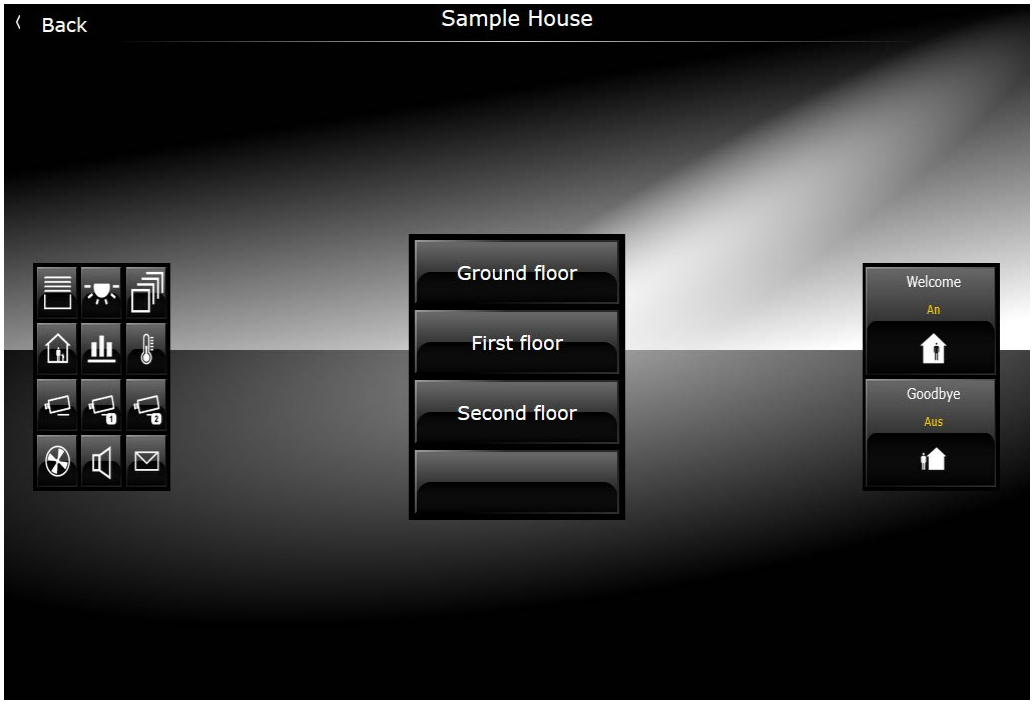
Process model: Select the existing process model that should be updated. Is the check mark at 'Generate Visu Pro projects automatically' set, the option is deactivated because the generator uses the file name of the Smart Assistant project.

Language: Choose between *German* and *English*.

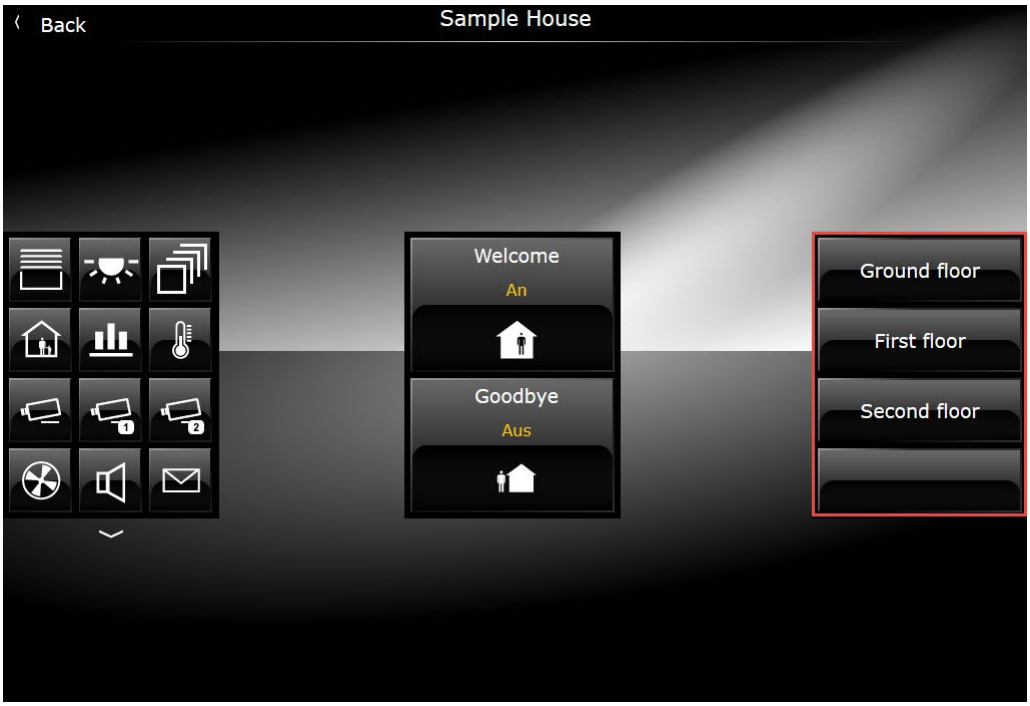
Resolution: Existing standard resolutions, set by the generator, are listed here.

Template: Template options change, depending on the resolutions. The following templates are available:

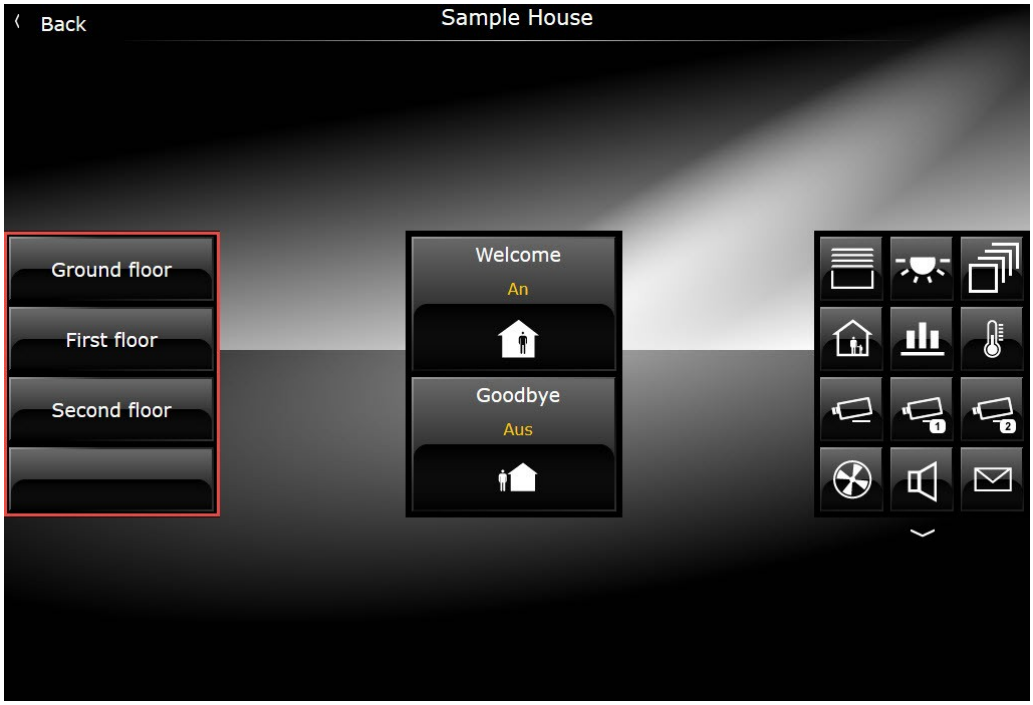
Standard:
Includes the mainpanel to switch between favourites, functions and rooms/floors. The following resolutions are available: 1024x768, 800x360, 1366x768, 768x1024



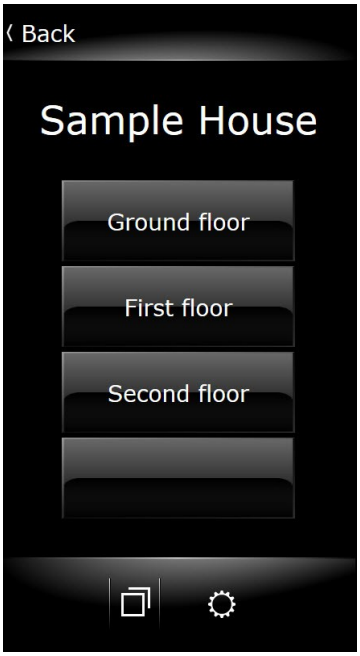
Tablet 1:
Template without mainpanel. Rooms/Floors will be presented in the right list. The following resolutions are available: 1024x768



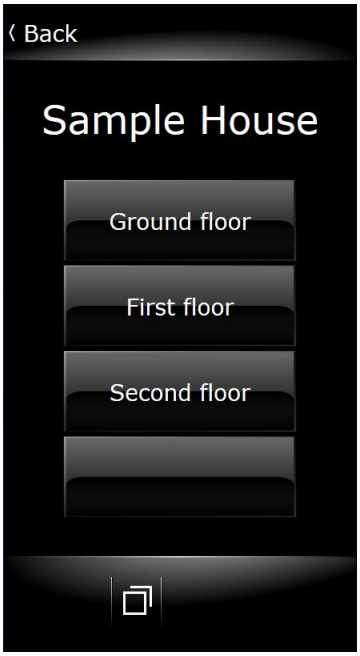
Tablet 2:
Template without mainpanel. Rooms/Floors will be presented in the left list. The following resolutions are available: 1024x768



Smartphone:
Template optimized for smartphones. Favourites can be accessed by clicking the toggle button. Click on the gearwheel to show the functions. The following resolutions are available: 440x800



Smart Control IP:
Template optimized for the Smart Control IP. Favourites can be accessed by clicking the toggle button. The following resolutions are available: 440x800



Delete existing worksheets: If the check mark is set the generator deletes all existing worksheets in the project before it generates the new ones from the Smart Assistant file.

Overwrite manual changes: If the project was already generated and than manually edited, the generator recognizes these changes since version 4.0 and does not overwrite them. Activating these option will overwrite the changes.

Generate level list: If this option is enabled in the project, lists of building floors will be generated and also the rooms will be divided after floors. This subdivision shall not be worth if you have buildings with only two floors and a manageable number of rooms, for example.

Generate control group list: This option is only usable, if the template for the smart control IP is selected. When you activate the option, every room gets an own list of element groups and for every element group a worksheet with the included elements.

2.2 Information about Generating

If you activate the option ***Generate Visu Pro Project automatically*** new projects will be generated for the KNX editor and the process model, otherwise if they already exist the KNX project and process model will be updated.

> **KNX editor:** Existing group addresses persist. If there's an overlap, the existing group addresses will be updated (name, type etc. are taken from the import file).
>
> **Process model:** Already existing functions will be updated. If previously a function has been renamed, the generator does not recognize this, so it will remain and a new one is created.

2.3 Update Project Lists

When the worksheet generator is started, the project lists are loaded again. Newly created projects are displayed automatically in the project selection lists. However, if the generator is already opened, the projects will not be automatically loaded.

Click the button ***Update*** to update the project lists.

JUNG Visu Pro
KNX Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Process Interface....3
1.4 Import from ETS....3
1.5 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....9

4 FUNCTIONS....9

4.1 KNX Project Organizer....9
4.1.1 Create KNX Project....10
4.1.2 Open KNX Project....11
4.1.3 Delete KNX Project....11
4.1.4 Rename KNX Project....12
4.2 Edit Project Properties....13
4.3 Group Addresses....14
4.3.1 Create Main Group....15
4.3.2 Create Middle Group....15
4.3.3 Create Group Address Range....16
4.3.4 Create Group Addresses....18
4.3.5 Edit Group Addresses....19
4.4 Start Groups....21
4.5 Request Groups....24
4.6 Report Groups....28
4.6.1 Report Formatting....33
4.7 KNX System Time....34
4.8 Synchronize PC Time with KNX Time....35
4.9 Application of KNX Data Types....41
4.10 ETS Import of KNX Group Addresses....44
4.10.1 Create ETS Group Addresses Print Report....45
4.10.2 Import OPC Export File from the ETS....45
4.10.3 Import of ETS Export Files into the KNX Editor....47
4.10.4 ETS-Import Settings....50
4.11 Import of Smart Assistant Address Files....52
4.12 Activate Address Scheme....52
4.13 KNX Settings....54
4.14 Connect to KNX Bus....56
4.15 Send Telegrams Manually....57
4.16 Message List....59
4.17 Logging KNX Messages into Archives....60
4.18 Data Cache Settings....61

5 REPORTS.....62
5.1 Create Report.....62
5.2 Print Preview.....64

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz
Main memory	1024 MB
Free disk space (additional disk space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Process Interface

The connection to the KNX bus requires the installation of a suitable version of the FALCON driver of the KNX Association and the appropriate interfaces.

Optionally, the visualization system is equipped with an OPC (OLE for Process Control) client, so that instead of or additionally to KNX, OPC servers - which are available for a multitude of automation systems – can be used for the connection to the process.

1.4 Import from ETS

ETS5	YES, use of ETS, project export
ETS4	YES, use of ETS, project export
ETS3, ETS2 version 1.3	YES, use of ETS, OPC export
ETS2 versions 1.1, 1.2 and 1.3	YES, use of print reports redirected to file
Earlier ETS versions	NO

1.5 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

After the installation of the **visualization system** a program group is created in the Start Menu. The default settings can be adopted or changed manually during installation.

Click the program symbol to start the control panel of the **visualization system**. In order to clearly distinguish between the different applications, different colors were used. The icon of the **KNX editor** contains a blue colored bar .

To start the individual function modules, double-click the symbol of the respective modules.

The **KNX editor** is part of the **visualization system**. It is used to create KNX projects which serve as a basis for the communication with the bus and for the operation of visualization projects.

First steps:

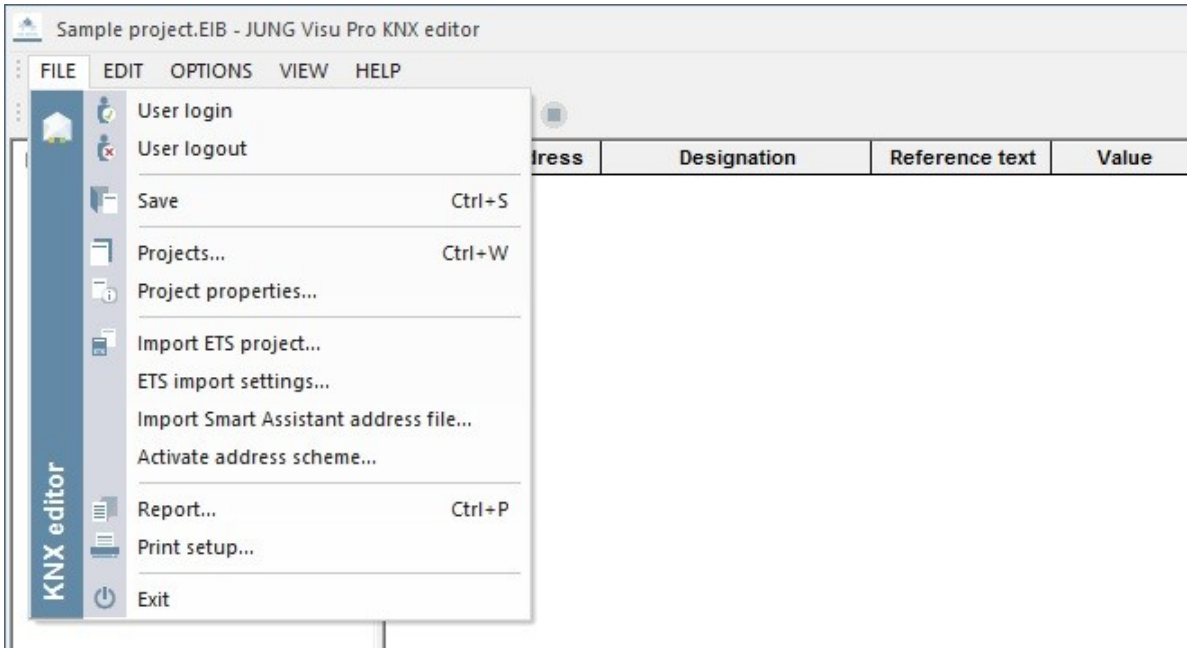
1. Create a KNX project.
2. Create group addresses manually or import them from an ETS project.
3. Specify information concerning the data points: data point type, unit and rights.
4. Save the settings.
5. Connect the project to the KNX bus.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

3 User Interface

3.1 Menu

Menu item *File*



User login: Opens a dialog to log in a user to the system.

User logout: Logs current user out of the system.

Save: Saves the project.

Projects: Opens a dialog to create and maintain KNX projects.

Project properties: Opens a dialog where you can edit general project properties.

Import ETS project: Opens a dialog to import an ETS project.

ETS import settings: Opens a dialog to set standard datapoint types that are used when the typ is not specified in the ETS import file.

Import Smart Assistant address file: Opens a dialog to import Smart Assistant address files.

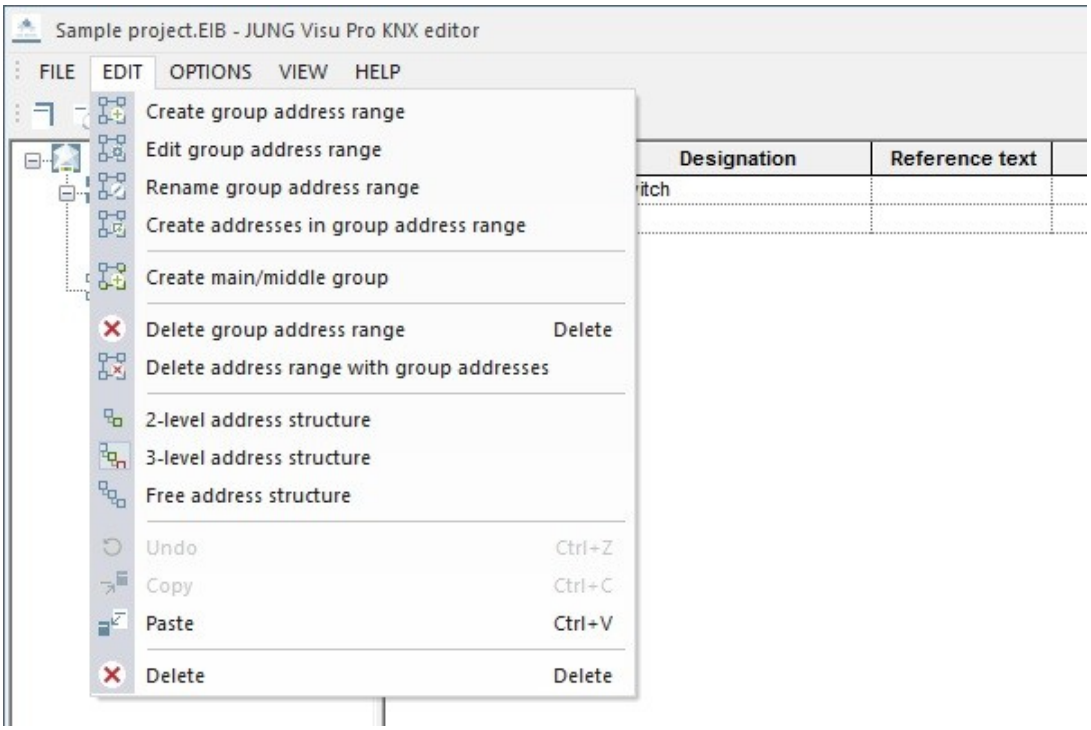
Activate address scheme: Opens a dialog to import an ETS project.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Quits the KNX editor. Shortcut ALT+F4.

Menu item *Edit*



Create group address range: Opens the dialog ***New address range***.

Edit group address range: Opens the dialog for editing the range limits of the selected group.

Rename group address range: Opens the input field of the selected group address range.

Create addresses in group address range: Opens the dialog ***Create group addresses***.

Create main/middle group: Opens the dialog ***New main group*** or ***New middle group***.

Delete group address range: Deletes only the selected address range. The group addresses contained in this address range will be placed in the address range above. If there is none, the group addresses will be deleted as well.

Delete address range with group addresses: Deletes the selected address range with all address ranges and group addresses included in that address range.

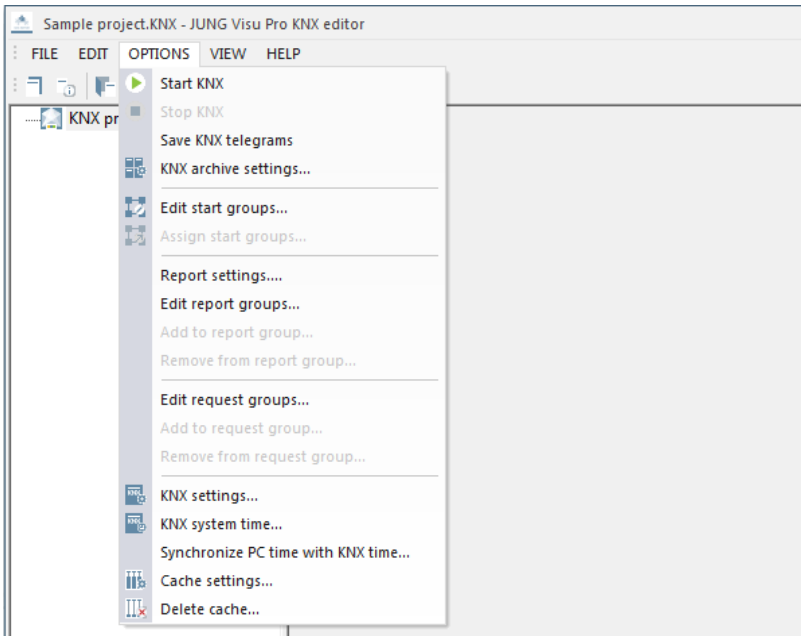
2-level address structure: Switches to the 2-level address structure for group addresses (00/0001).

3-level address structure: Switches to the 3-level address structure for group addresses (00/0/001).

Free address structure: Switches to the free address structure for group addresses (00001).

Standard editing options: Undo, Copy, Paste, Delete.

Menu item *Options*



Start/stop KNX: Connects to or disconnects from the KNX bus.

Save KNX telegrams: Saves KNX telegrams to a file.

KNX archive settings...: Opens a dialog to edit archive settings.

Edit start groups...: Opens a dialog to edit start groups.

Assign start groups...: Opens a dialog to assign start groups.

Report settings: Opens the dialog to edit the report groups.

Edit report groups: Opens the dialog to edit report groups.

Add to report group: Opens the dialog for the selection of report groups.

Remove from report group: Opens the dialog for the selection of report groups.

Edit request groups: Opens the dialog to edit request groups.

Add to request group: Opens the dialog for the selection of request groups.

Remove from request group: Opens the dialog for the selection of request groups.

KNX settings: Opens a dialog to configure the connection to KNX.

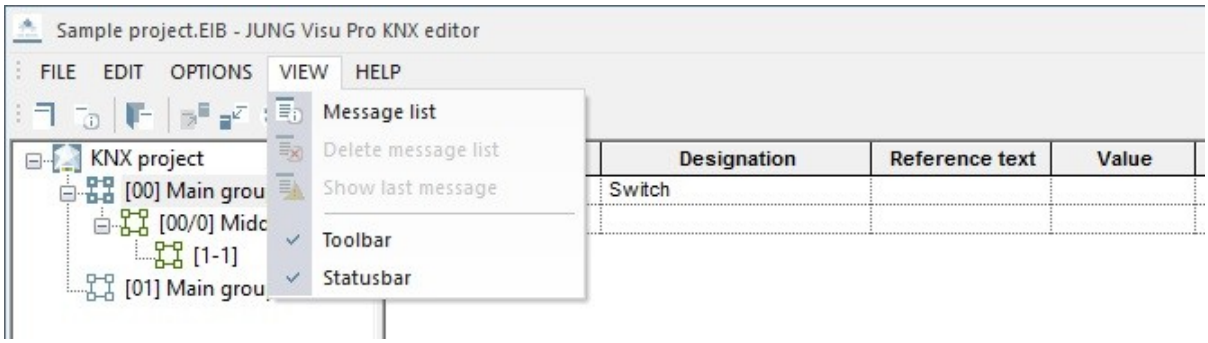
KNX system time: Opens a dialog to update the KNX clock from KNX.

Synchronize PC time with KNX time: Opens a dialog to synchronize the PC time with KNX time.

Cache settings: Opens a dialog to configure the data cache.

Delete cache: Deletes the cache.

Menu item *View*

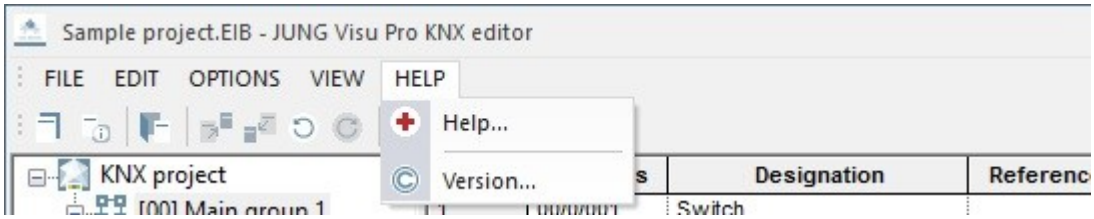


Message list: Toggles the visibility of the messages list at the bottom of the editor.

Delete message list: Deletes the messages from the message window. This operation does not affect the saving of telegrams to an archive file.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

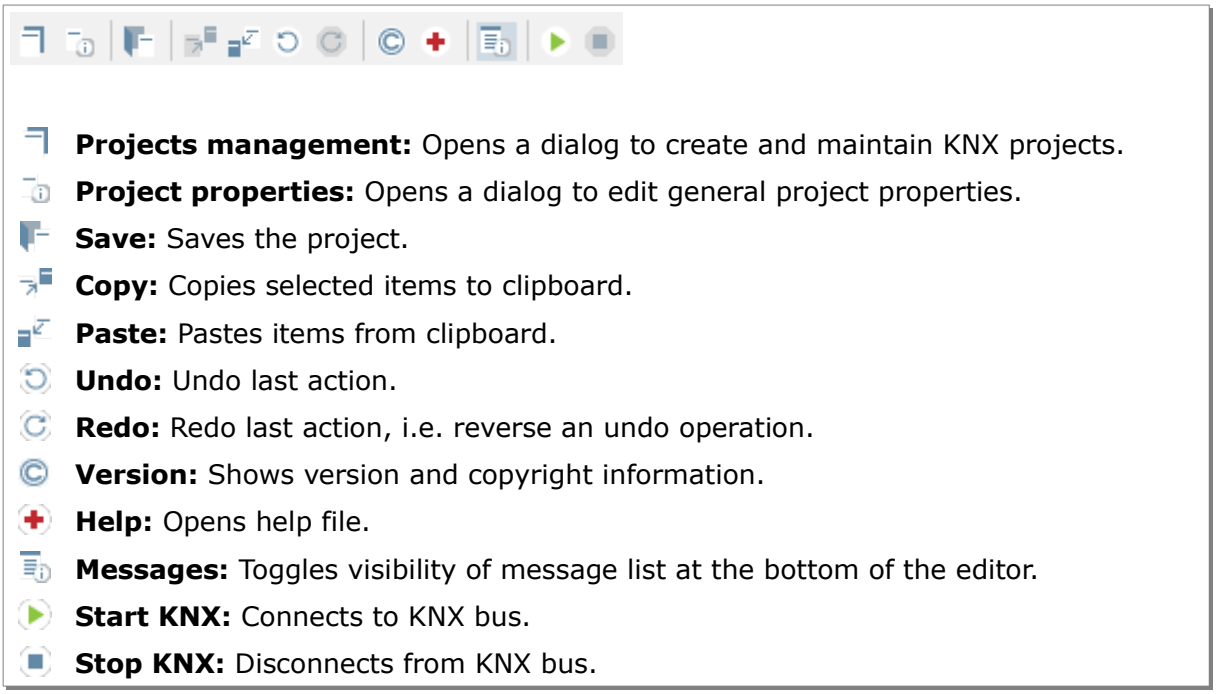
Menu item *Help*



Help: Opens help file.

Info: Shows version and copyright information.

3.2 Toolbar



Projects management: Opens a dialog to create and maintain KNX projects.
Project properties: Opens a dialog to edit general project properties.
Save: Saves the project.
Copy: Copies selected items to clipboard.
Paste: Pastes items from clipboard.
Undo: Undo last action.
Redo: Redo last action, i.e. reverse an undo operation.
Version: Shows version and copyright information.
Help: Opens help file.
Messages: Toggles visibility of message list at the bottom of the editor.
Start KNX: Connects to KNX bus.
Stop KNX: Disconnects from KNX bus.

4 Functions

4.1 KNX Project Organizer

In the dialog ***KNX Project Organizer*** you can create new projects and open, close, delete or rename existing projects.

In the project organizer you can use the buttons on the sidebar or the commands under the menu item ***Project***.

The project organizer provides the following options:

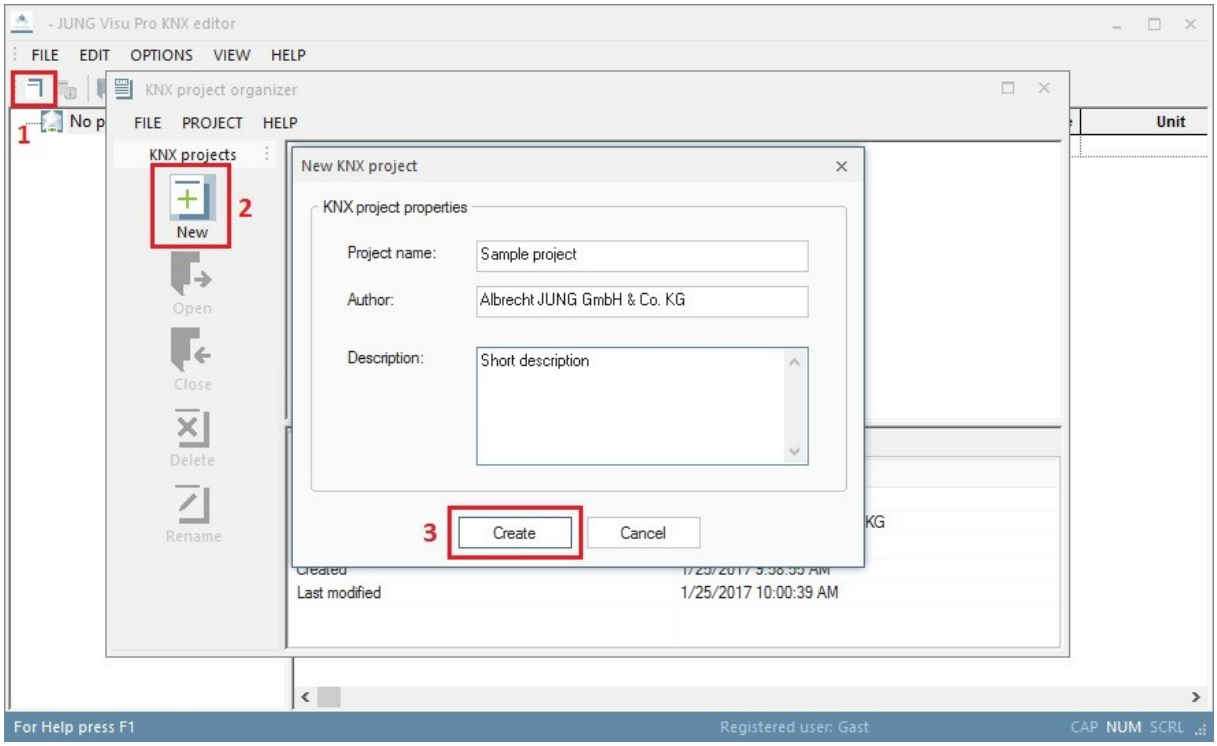
- **New**: Create a new KNX project.
- **Open**: Open an existing project in the KNX editor.
- **Close**: Close the currently opened KNX project.
- **Delete**: Delete the selected project.
- **Rename**: Rename a project.

4.1.1 Create KNX Project

The dialog ***KNX project organizer*** opens automatically when the KNX editor is started.

To start it manually in the KNX editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

To create a new KNX project, click the button ***New*** on the sidebar or choose the menu item ***Project*** and click ***New***. The dialog ***New KNX project*** opens.



Enter the data of the new project:

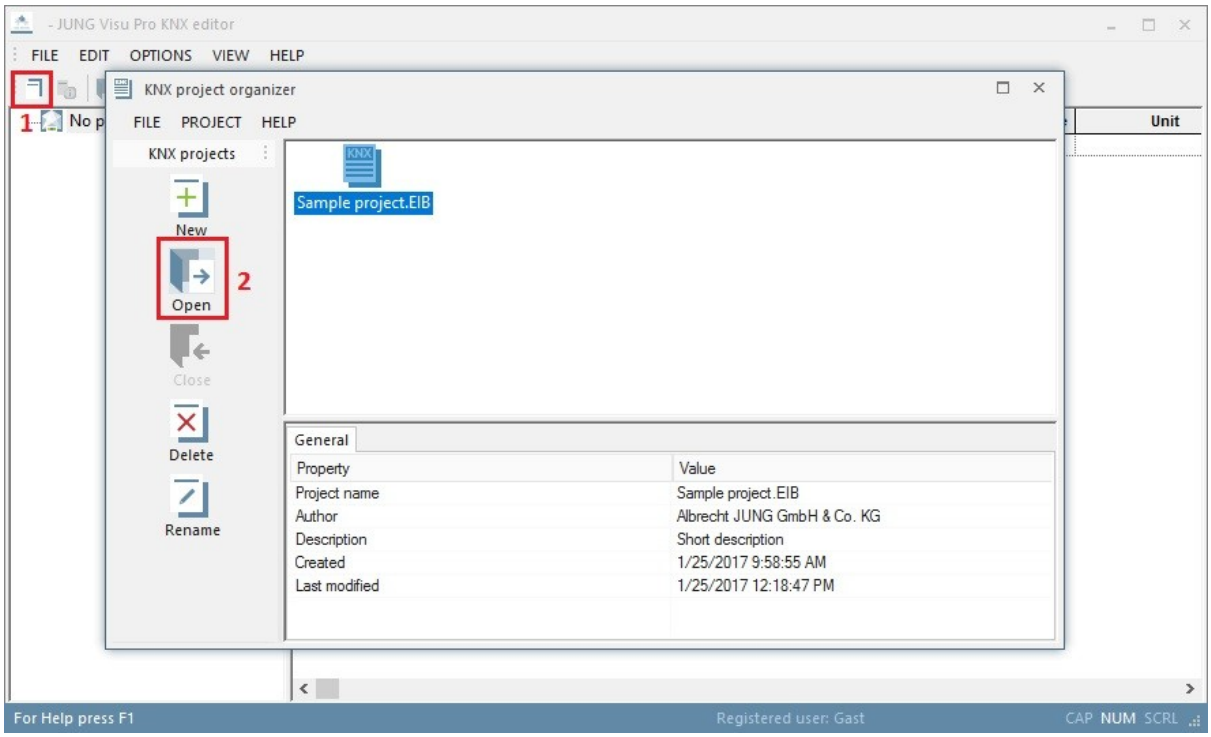
Project name: Enter a unique name for the KNX project.

Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or edited later.

Create: Press the button ***Create*** to commit the entries and to finish the creation of the project.

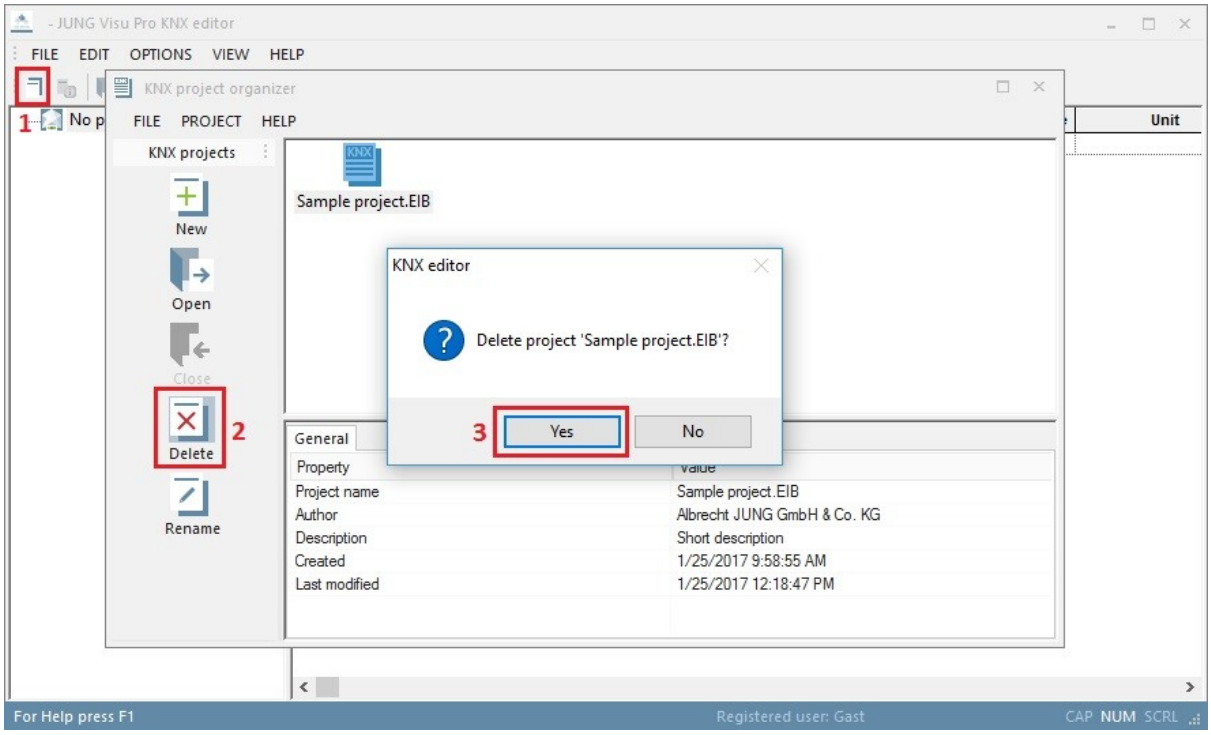
4.1.2 Open KNX Project

To load a KNX project into the KNX editor, double-click the project symbol in the project list or select the desired project and click the button ***Open*** in the sidebar.



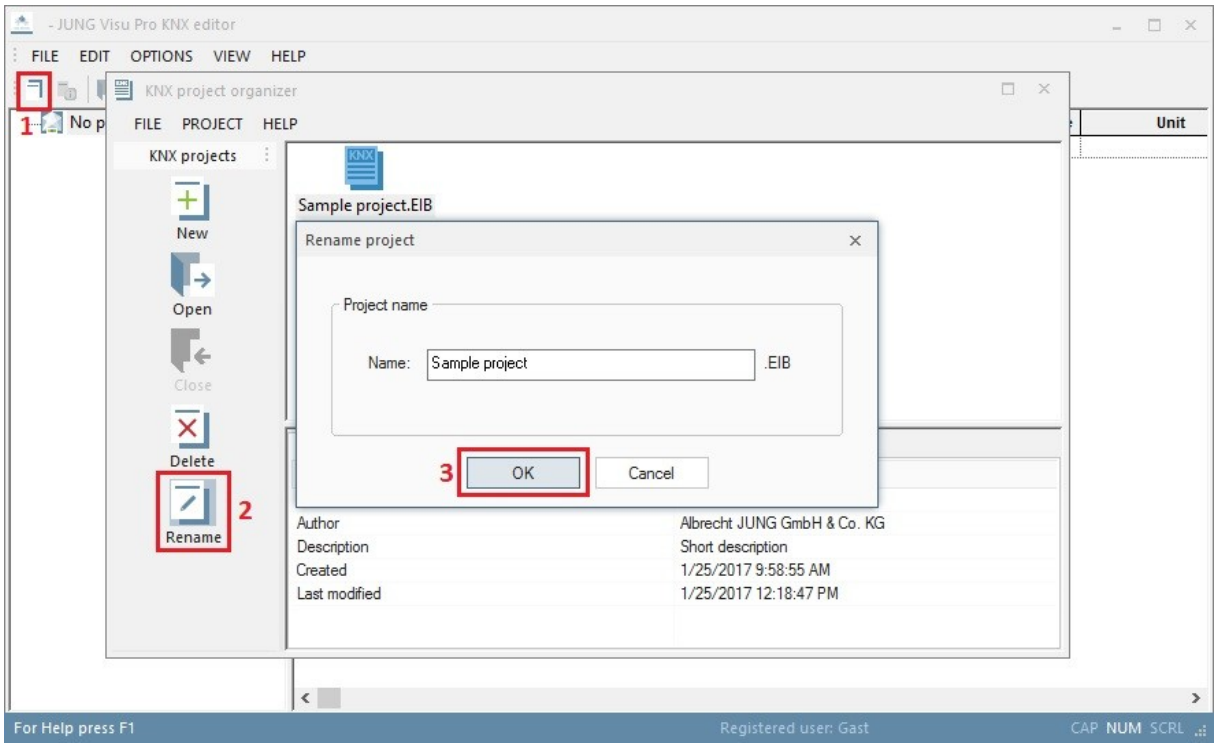
4.1.3 Delete KNX Project

To delete a project, select the project and click the button ***Delete*** in the sidebar.



4.1.4 Rename KNX Project

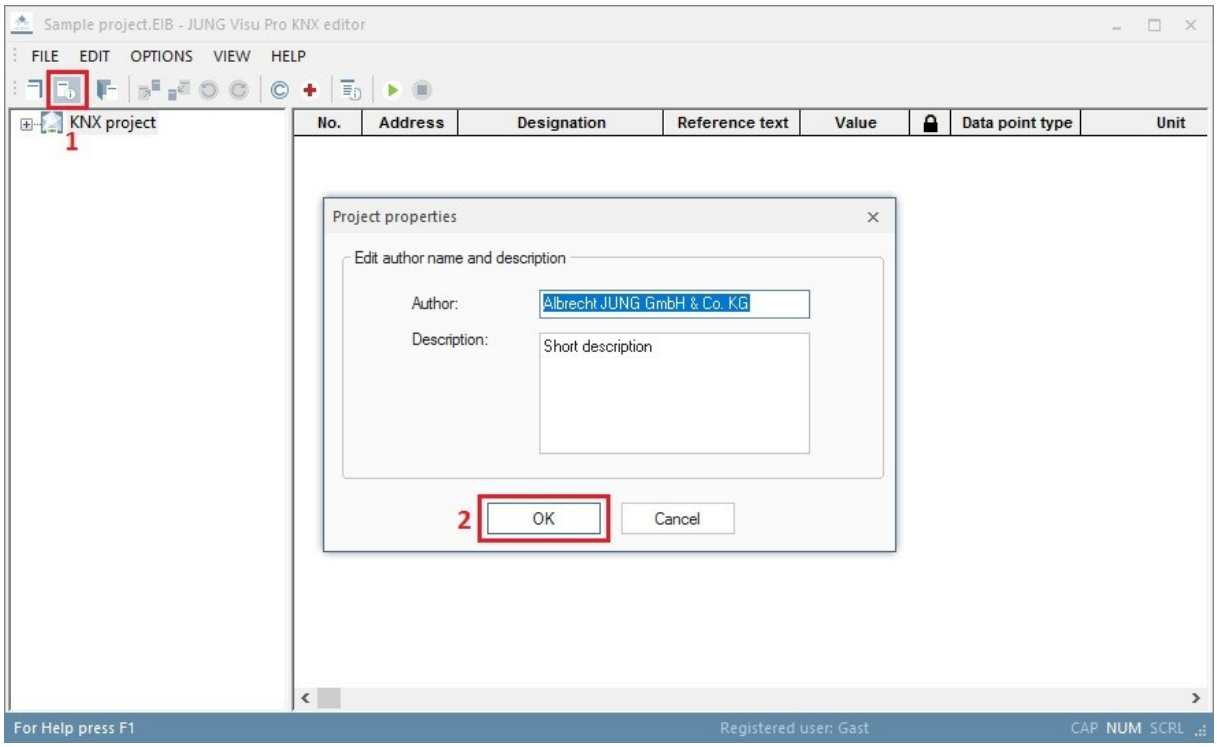
To rename a project, select the desired project and click the button ***Rename*** in the sidebar. The dialog ***Rename project*** opens.



Enter in the name field the new project name and click the button ***OK***. The dialog is closed and the new project name is adopted. The extension ***.EIB*** is automatically added to the project name.

4.2 Edit Project Properties

Choose in the KNX editor the menu item ***File*** and click ***Project properties*** or click the symbol in the toolbar. The dialog ***Project properties*** opens where you can edit general properties of the project.

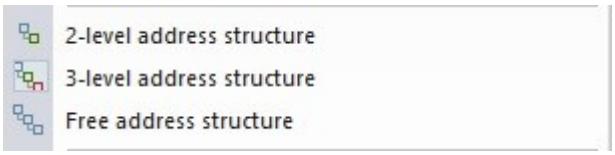


Edit the specifications author and description and click the button ***OK*** to commit the changes and to close the dialog.

4.3 Group Addresses

You can create group addresses manually or import them from ETS projects via the import function. More details on the ETS import will be provided in one of the subsequent chapters. In the following, the manual creation of group addresses is described. Manually created group addresses can not be exported to the ETS.

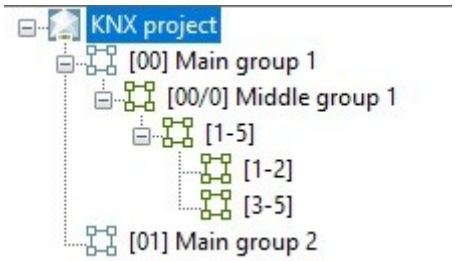
Under the menu item ***Edit*** you can choose between 3 different structures of group addresses:



2-level address structure: The group addresses are arranged in a 2-level scheme; main group address and group address *(00/0001)*.

3-level address structure: The group addresses are arranged in a 3-level scheme; main group address, middle group address and group address *(00/0/001)*.

Free address structure: Free display of the group addresses *(00001)*.

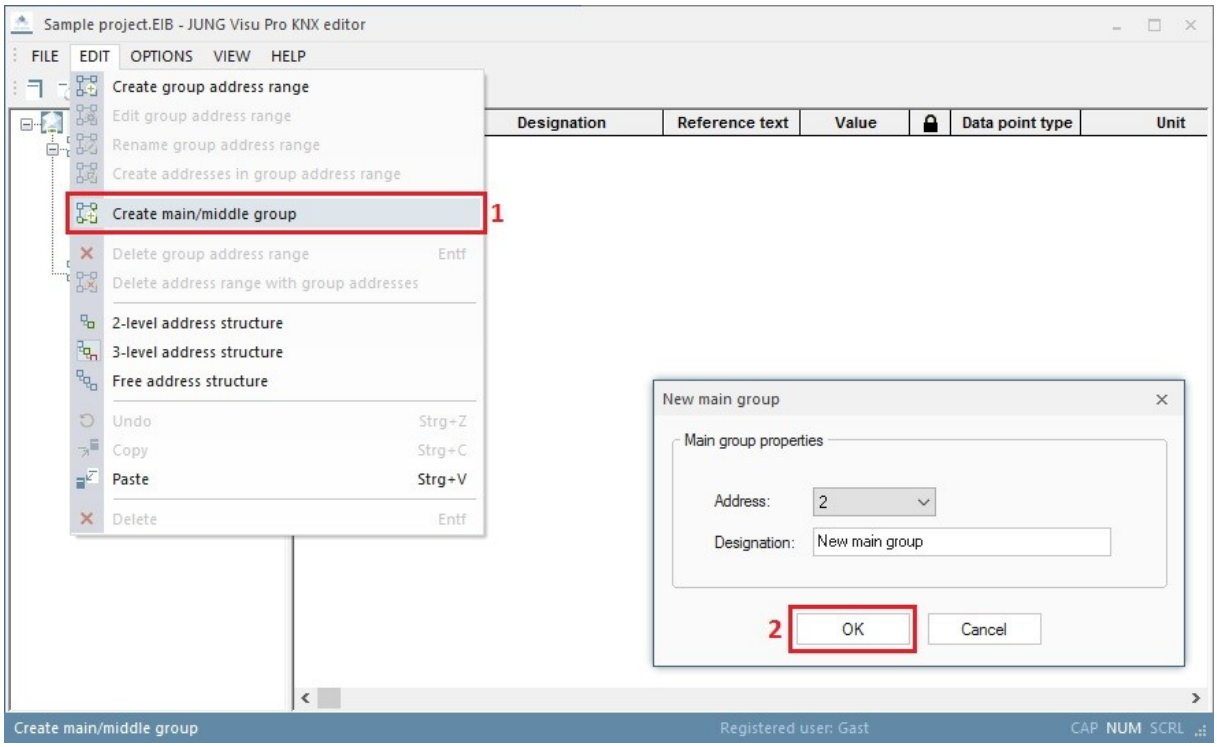


In the left window of the editor you can see the KNX projects with their respective groups (if already created) arranged in a structure tree.

4.3.1 Create Main Group

To create a main group,

- choose the menu item ***Edit*** and click ***Create main/middle group*** or
- right-click the project symbol in the directory tree and click in the context menu the option ***Create main/middle group***.



The dialog ***New main group*** opens. Enter the main group address and a name for the main group. Click the button ***OK*** to commit the data and to close the dialog.

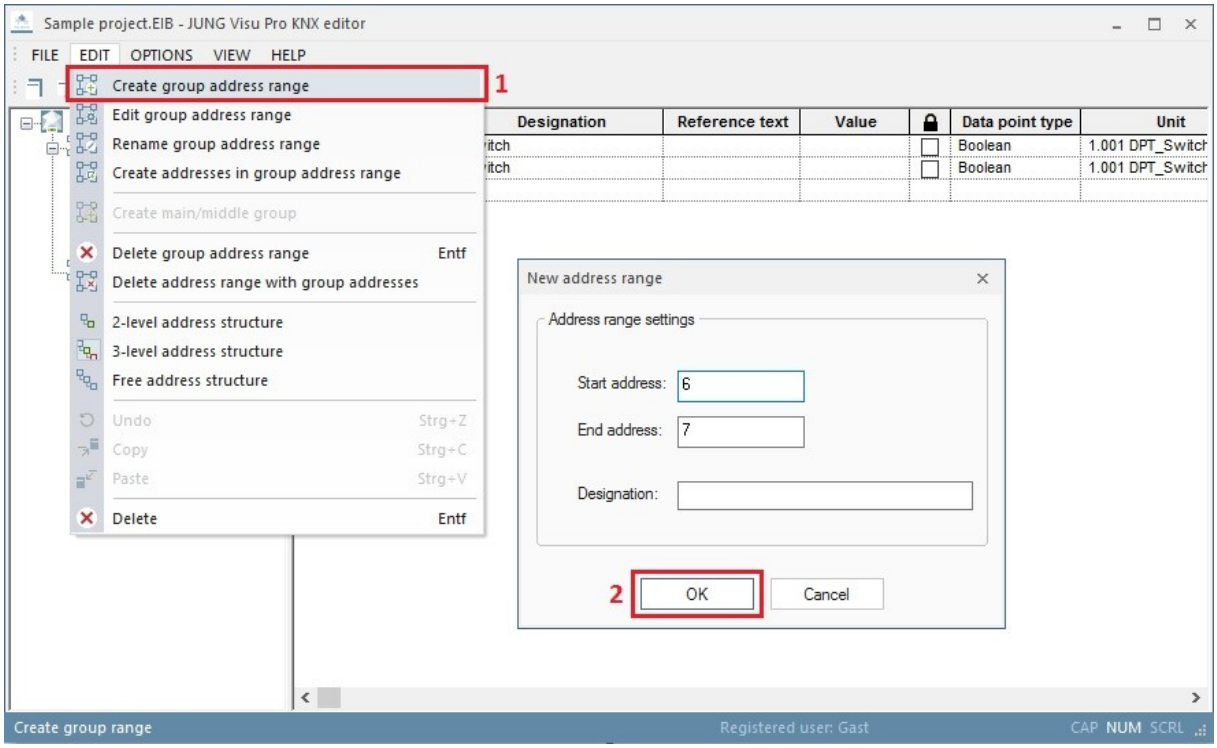
Use this method to create further KNX main group addresses.

4.3.2 Create Middle Group

To create a middle group, choose the menu item ***Edit*** and click ***Create main/middle group***. Alternatively, right-click the main group symbol in the directory tree and click in the context menu the option ***Create main/middle group***.

4.3.3 Create Group Address Range

To create a group address range, select in the directory tree the project or group for which you want to create the address range. Choose the menu item ***Edit*** and click ***Create group address range***. Alternatively, right-click the project or group in the directory tree and click in the context menu the option ***Create group address range***.



The dialog ***New address range*** opens. Enter the start and the end address (range limits) and, where appropriate, a name for the selected group address range. Click the button ***OK*** to commit the entry.

The specified address range will then be assigned to the corresponding or the superordinate address range.

Change group address range:

Select in the directory tree the group address range you wish to edit. Choose the menu item ***Edit*** and click ***Edit group address range***. A dialog opens where you can change the range limits of the selected group address range. Enter the new range limits and click the button ***OK*** to commit the change.

Rename group address range:

Select in the directory tree the group address range you wish to rename. Choose the menu item ***Edit*** and click ***Rename group address range***. The input field of the selected group address range opens. Enter the new name and press the **enter** key.

Delete group address range:

Select in the directory tree the group address range you wish to delete. Choose the menu item ***Edit*** and click ***Delete group address range***. Answer the confirmation prompt with ***Yes*** to delete the selected address range. The contained ranges and group addresses will be placed in the superordinate group address range. If there is no superordinate range, the group addresses will be deleted as well.

Delete group address range with group addresses:

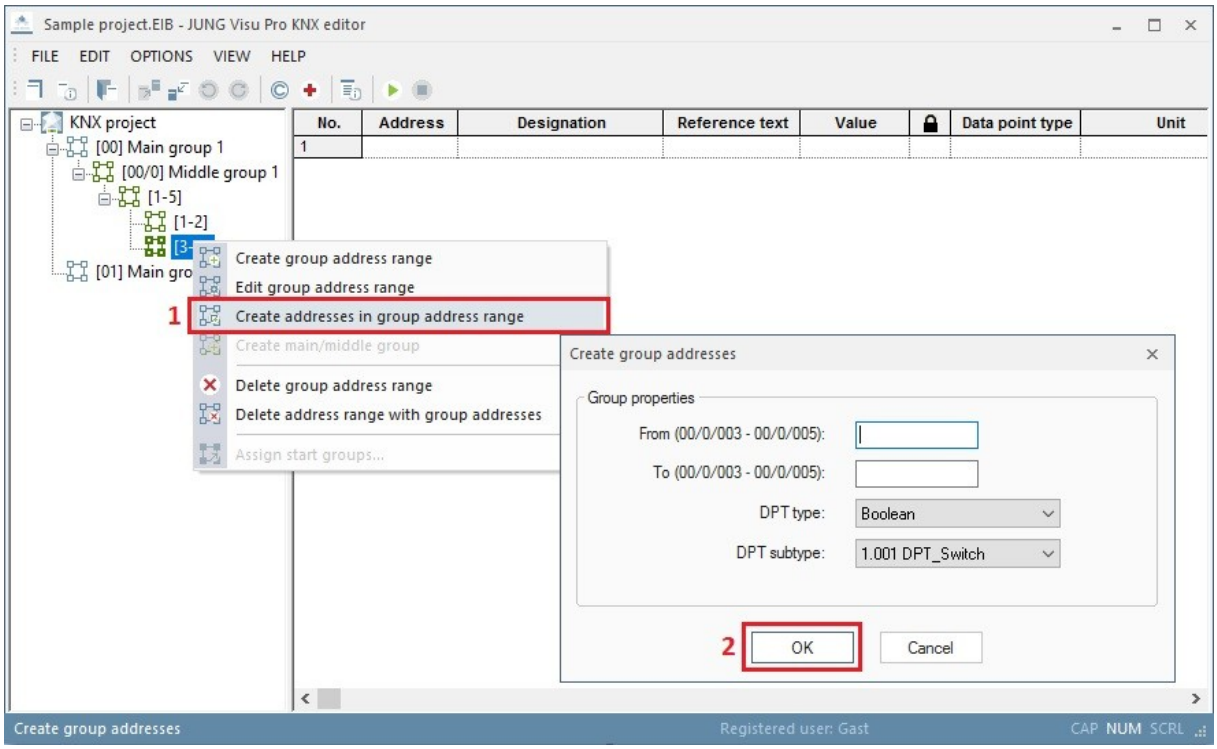
Select the group address range in the directory tree. Choose the menu item ***Edit*** and click ***Delete address range with group addresses***. Answer the confirmation prompt with ***Yes*** to delete the selected address range along with all ranges and group addresses contained in it.

4.3.4 Create Group Addresses

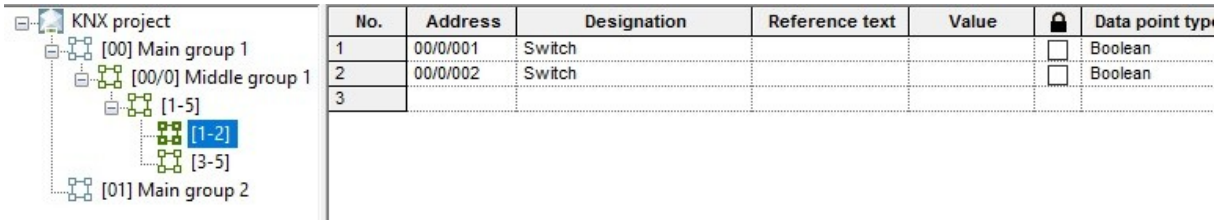
To create group addresses in a group address range,

- select the group address range, choose the menu item ***Edit*** and click ***Create addresses in group address range*** or
- right-click the group address range and click in the context menu the option ***Create addresses in group address range***.

The dialog ***Create group addresses*** opens.



The entry fields ***From*** and ***To*** allow to enter several group addresses in the selected address range, having the same data point type. If you already know the number of the required group addresses, you can create them all at once and configure them manually in the fields afterward. Here you can also change the middle group address.



4.3.5 Edit Group Addresses

Each group address requires further specifications to enable the KNX editor to properly interpret KNX telegrams:

Address: You can enter an address directly in the address column. Alternatively, select the group address range in the directory tree, right-click it and click in the context menu the option ***Create addresses in group address range***.

Designation:

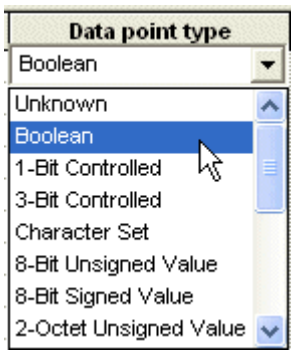
Enter a name for the group address. The designation serves the identification of the address during further processing of the data. This is a mandatory field! The KNX editor and other parts of the system will refer to the KNX group address by this name.

No.	Address	Designation
1	00/0001	Light 1

Value: If a group address receives values, they are displayed in this field. If write access is allowed, you may enter a value in this field and send a value to the KNX group address.

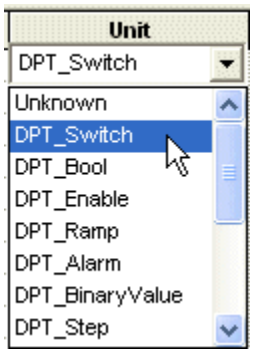
Reference text: An alternative text for the installation name which will be exported to the csv file as well.

Lock address: When this option is enabled, the data point will not be updated during an ETS import.



Data point type:

Type of the KNX address. To assign a data point type (DPT), click in the input field and select the desired data point type.



Unit:

After a type is set, an applicable KNX unit can be chosen. The specification of the unit is **optional**. If the unit has been defined for a group address, it will be shown in the messages list in a plain text message.

To assign a unit, click in the input field and select a unit from the dropdown list. The list automatically contains only the units which can be assigned to the data point type.

Rights:

To execute the desired action via the bus, the read and write rights must be set.

None: This is the default: Neither read nor write access is allowed. It is not possible to read the KNX group address nor to write values to it. However, spontaneous messages from this KNX group address can be received and interpreted.

Read: It is only possible to send read commands to the group address (no write commands).

Write: It is only possible to send a value to the KNX group address. The KNX editor denies read access to the address (important for e.g. switch objects).

Read/Write: It is possible to send a value to the group address and to read it.

Initialization: Here you can assign the group address to a start group.

Description: Enter a short description of the address (optional).

Ref.Unit (0): The notation of a certain state can be changed here (for example if an "off" instead of 0 is expected). This will be exported with the report data.

Ref.Unit (1): The notation of a certain state can be changed here (for example if an "on" instead of 1 is expected). This will be exported with the report data.

Note: *When you move the cursor to another row, either with the mouse or keyboard keys, the input of the current KNX group address will be committed. Leaving the edited row will commit the row's data!*

A new empty row will be displayed to enter the next KNX group address.

The cursor can be moved either with mouse click on a cell or keyboard arrow keys. The field under the cursor can be edited. To move the cursor between the cells of one row, use the <tab> and <shift>+<tab> keyboard keys.

Save entries:

Before the new group address definitions can be used, the data must be saved. The editor can only interpret telegrams from the KNX bus after the newly entered data have been saved.

To save the project, choose the menu item ***File*** and click ***Save*** or click the symbol in the toolbar.

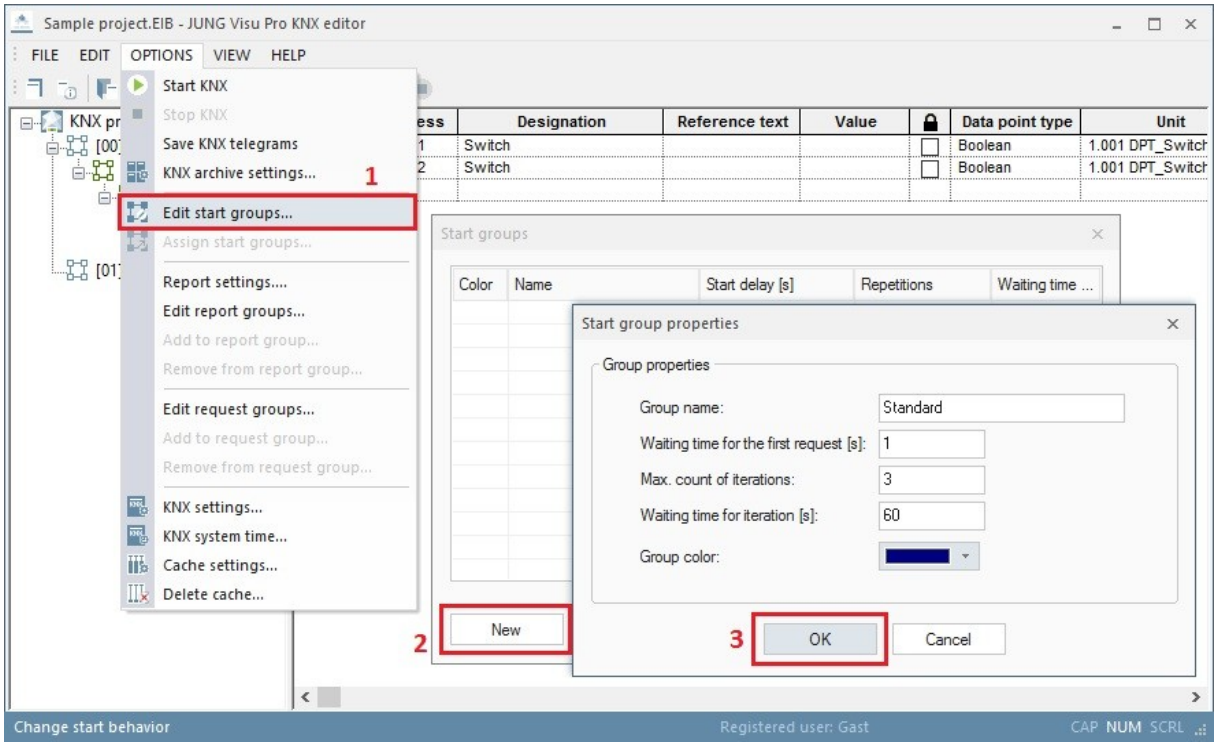
Warning: *Only after saving will the changes be interpreted by the bus!*

4.4 Start Groups

Through the **start groups**, group addresses can be specifically initialized. You can enter a waiting time to specify when a value should be initialized after the start of the project.
It is recommended to use this feature only for status variables.

Choose the menu item ***Options*** and click ***Edit start groups***. The dialog ***Start groups*** opens.

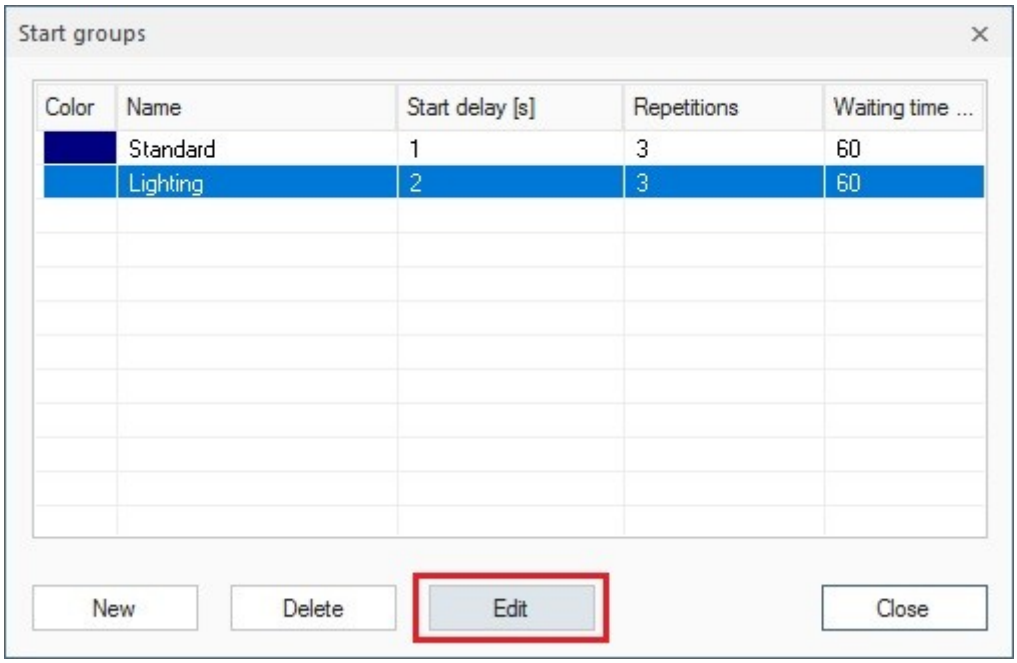
Create start group:



Click the button ***New***. The dialog ***Start group properties*** opens. Enter the settings and press the button ***OK*** to create the start group and to close the dialog. The newly created start group is now available in the dialog ***Start groups***.

Edit start group:

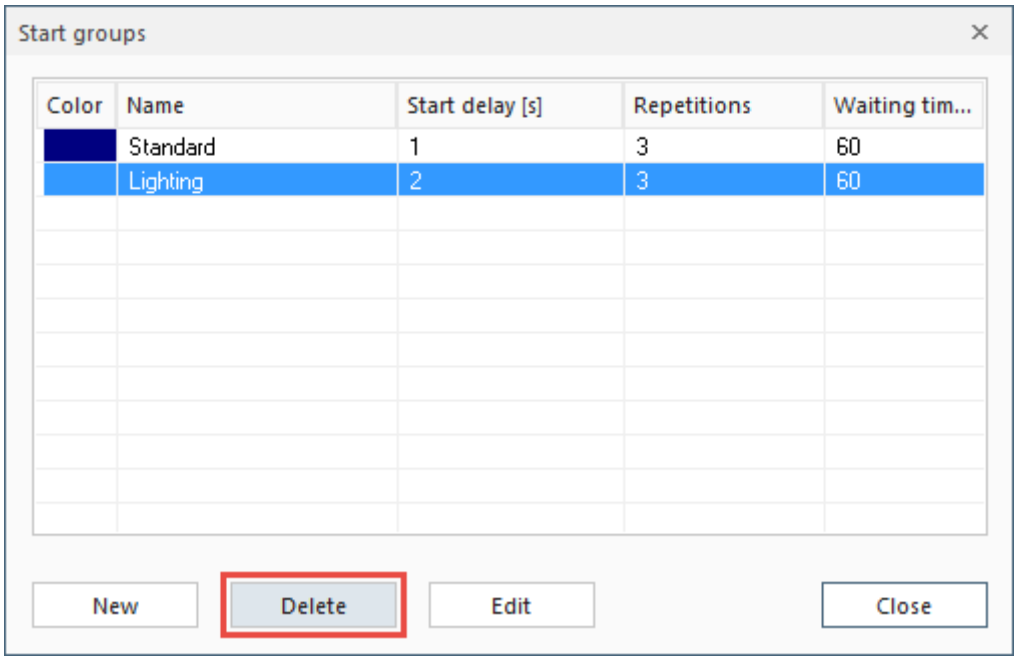
To edit the start group properties, select the start group and click the button ***Edit***. The dialog ***Start group properties*** opens.



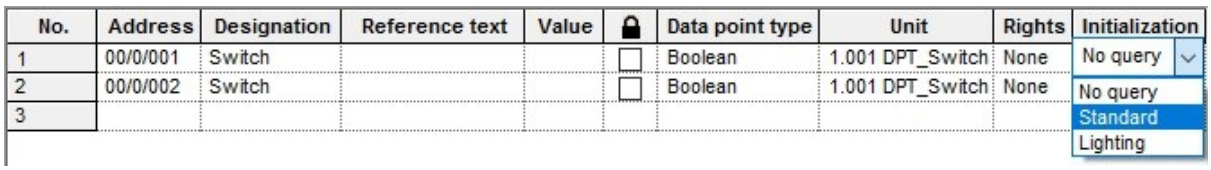
Edit the start group properties and click the button ***OK*** to commit the entries.

Delete start group:

To delete a start group, select the start group and click the button ***Delete***. Then click the button ***Close*** to quit the dialog ***Start groups***.

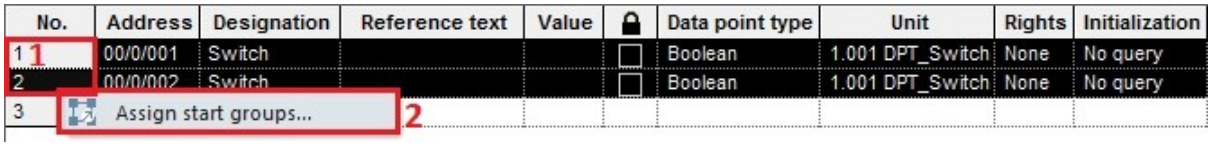


Assign start group:

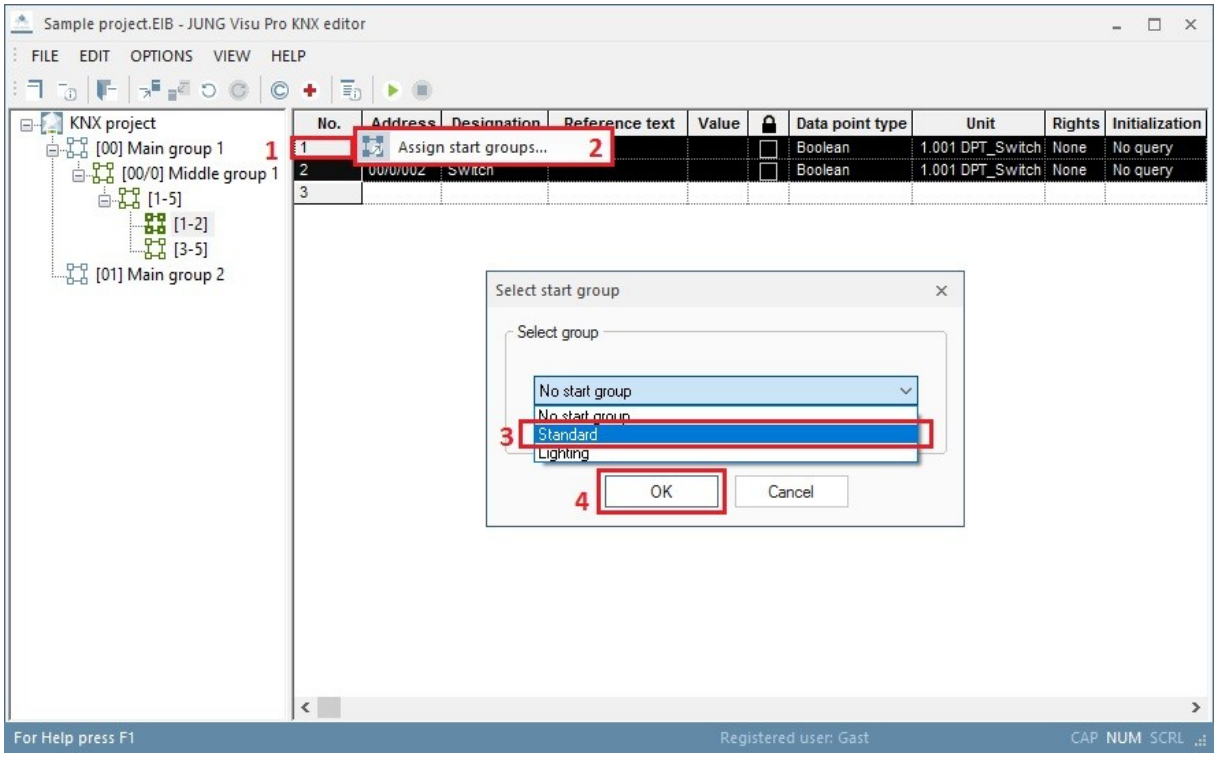


To assign a start group to a group address, click in the column ***Initialization***. Then select the start group in the combo box.

You can assign multiple group addresses to the same start group: Select the desired group addresses while pressing the control key in the first column of the schedule. Right-click to open the context menu and choose ***Assign start groups...*** The dialog ***Select start group*** opens.



Or: To assign all group addresses to a start group, click on the top left cell of the grid to select the whole table. Then right-click to open the context menu and click ***Assign start groups***. The dialog ***Select start group*** opens.



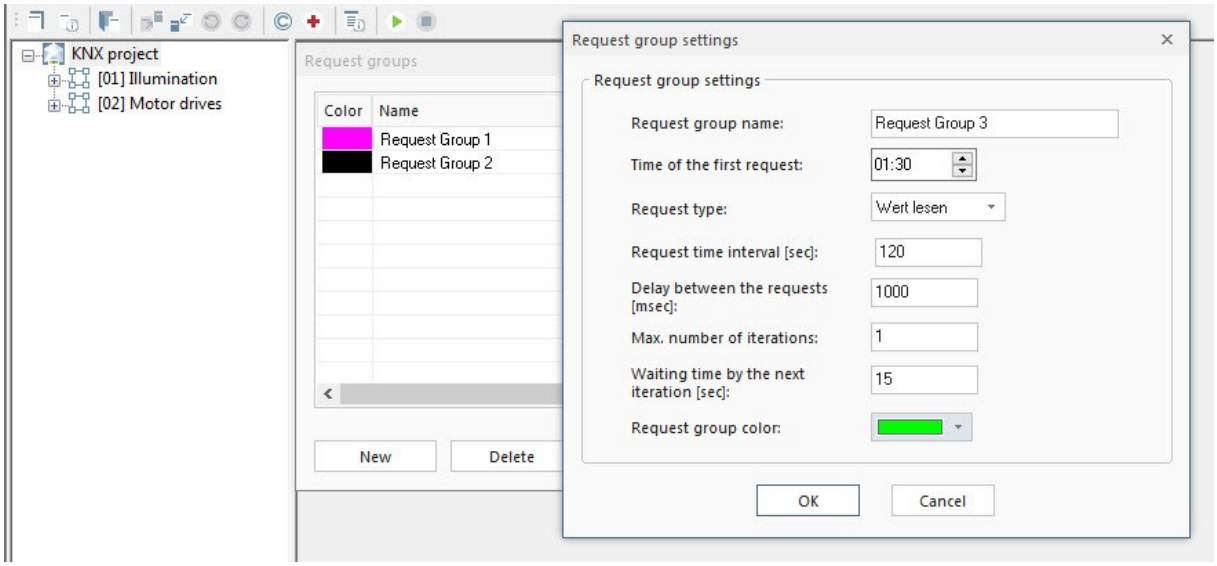
Select the desired start group and click ***OK*** to commit the entry.
All group addresses have now been assigned the start group ***Standard***.

4.5 Request Groups

Through the **request groups**, values can be read frequently. You can enter periodically when a value should be read or you can send 1 or 0 (if the end device needs a request value before sending back its value).

Click under ***Options*** at ***Edit request groups*** to open the dialog ***Request groups***.

Create request group:



Click at ***New*** to open the dialog ***Request group settings***. Insert the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Request groups.***

Request group name: Name of the group with which the group is listed.

Time of the first request: Time between 00:00 and 23:59.

Request type: Read value / Send 1 / Send 0.

Request time interval [sec]: Time specification between 10 and 86400 seconds.

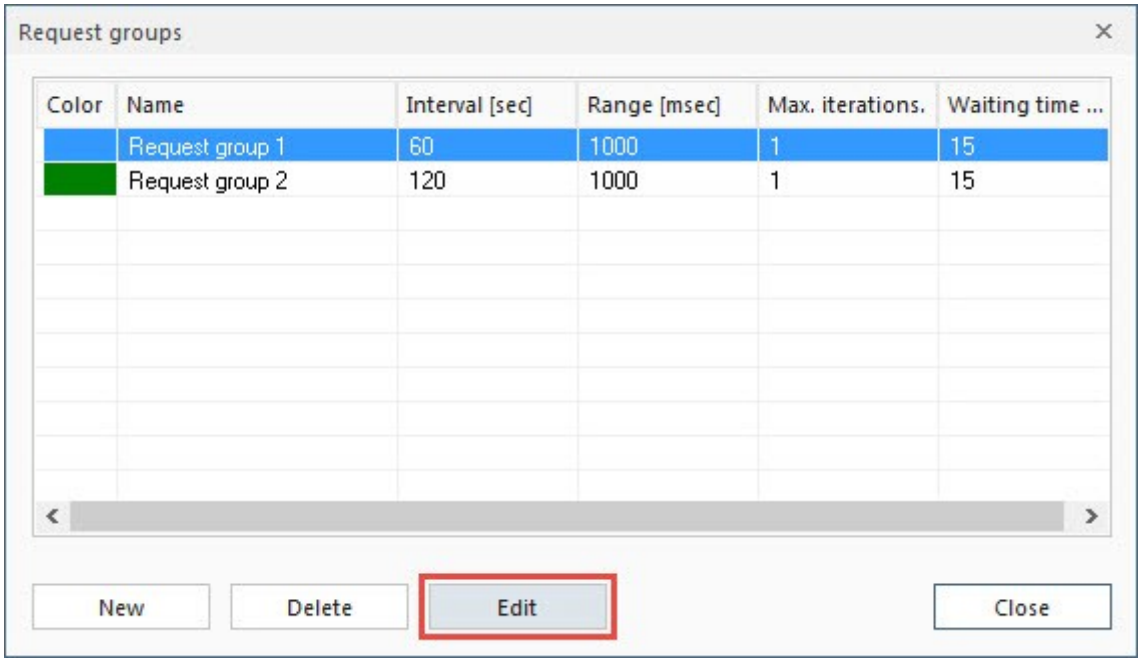
Delay between the requests [msec]: Time specification must be a value between 250 and 60000 milliseconds. This option determines the break between different request groups (necessary at a project with many data points to not overcharge the bus and lose protocols).

Max. number of iterations: Determines the number of repeats if the request failed.

Waiting time by the next iteration[sec]: Determines the time between the repeats.

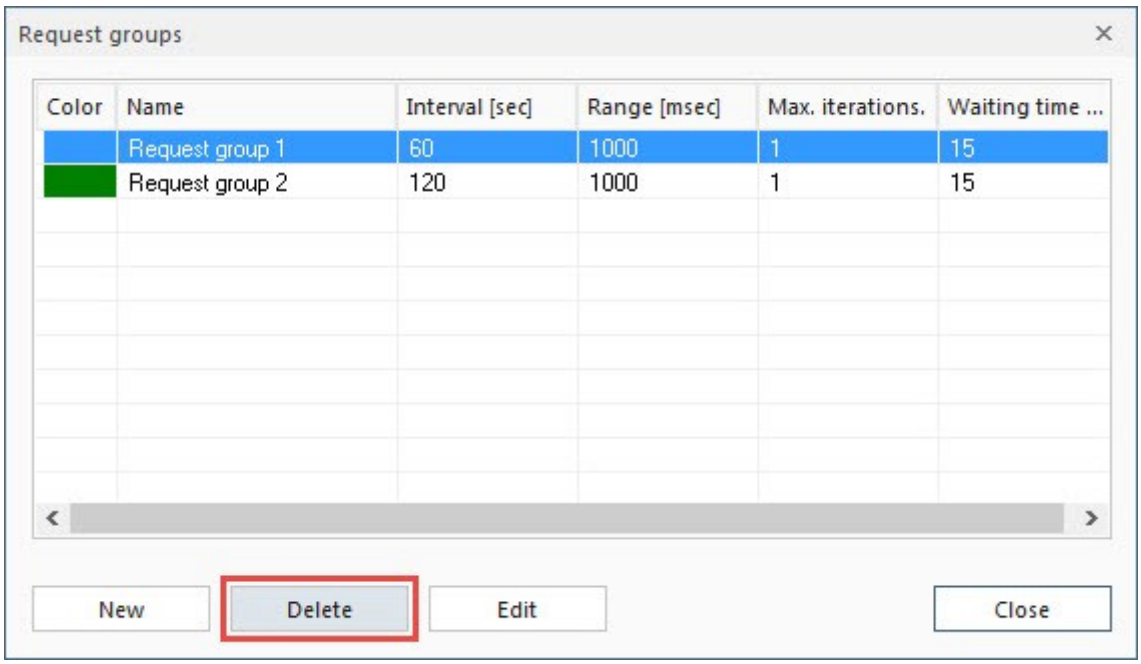
Request group color: Set a color to have a better discrimination of the groups.
Edit request group:

To edit a request group, select the request group and click the button ***Edit***. The dialog ***Request group settings*** opens.



Edit the group settings and save the changes by clicking ***OK***.

Delete request group:



To delete a request group, select the request group and click the button ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Request groups***.

Assign request group:

No.	Unit	Rights	Initialization	Request groups
1	1.001 DPT_Switch	Read/Write	No query	No request group
2	1.001 DPT_Switch	Read/Write	No query	Request group 1
3	1.001 DPT_Switch	Read/Write	No query	Request group 2

To assign a request group to a group address, click in the column ***Request groups*** to select one or multiple request groups.

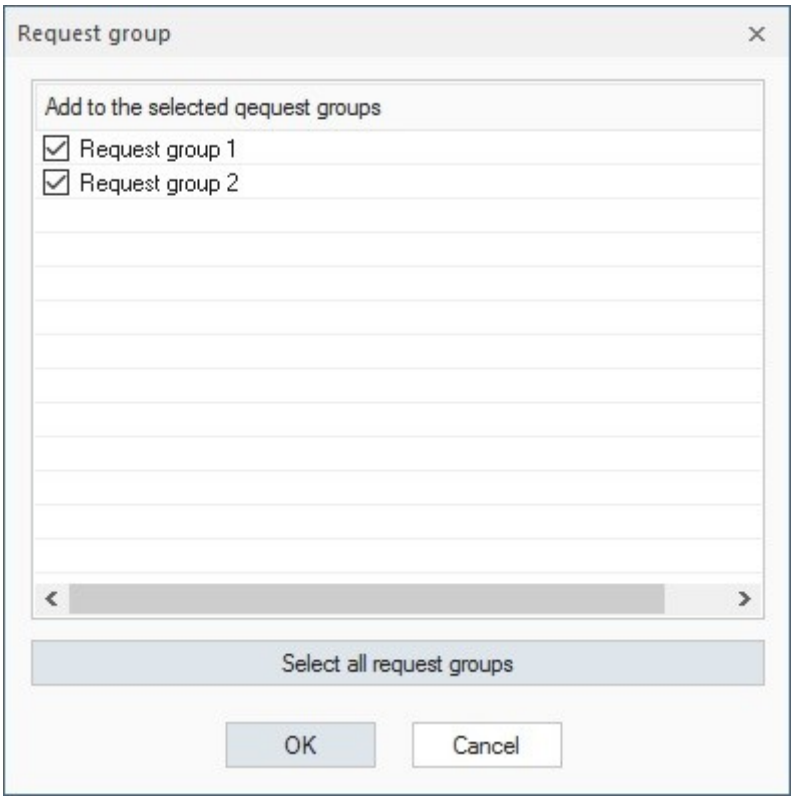
Or: To assign a selection of group addresses to a request group, select the first group address in the column ***Request groups*** and draw the mouse over the beneath lying lines. Right-click on the selected elements to open the context menu and choose ***Add to request group***.

No.	Rights	Initialization	Request groups	Report groups	Ref.unit (0)
1	Read/Write	No query	No request group	No report grou	
2	Read/Write	No query	No request group	No report grou	
3	Read/Write	No query	No request g		
4	Read/Write	No query	No request g		
5	Read/Write	No query	No request g		
6	Read/Write	No query	No request group		

Add to request group...

Remove from request group...

The dialog ***Request group*** opens.



Set a checkmark at the desired request groups and confirm with ***OK***.

No.	Unit	Rights	Initialization	Request groups
1	1.001 DPT_Switch	Read/Write	No query	2 request groups
2	1.001 DPT_Switch	Read/Write	No query	2 request groups
3	1.001 DPT_Switch	Read/Write	No query	Request group 2
4	1.001 DPT_Switch	Read/Write	No query	Request group 1
5	1.001 DPT_Switch	Read/Write	No query	Request group 1
6	1.001 DPT_Switch	Read/Write	No query	Request group 2

For every group address is the request group now visible or, if a group address has multiple request groups, the number of request groups.

Delete request group:

Deleting request groups is analog to assigning request groups.

4.6 Report Groups

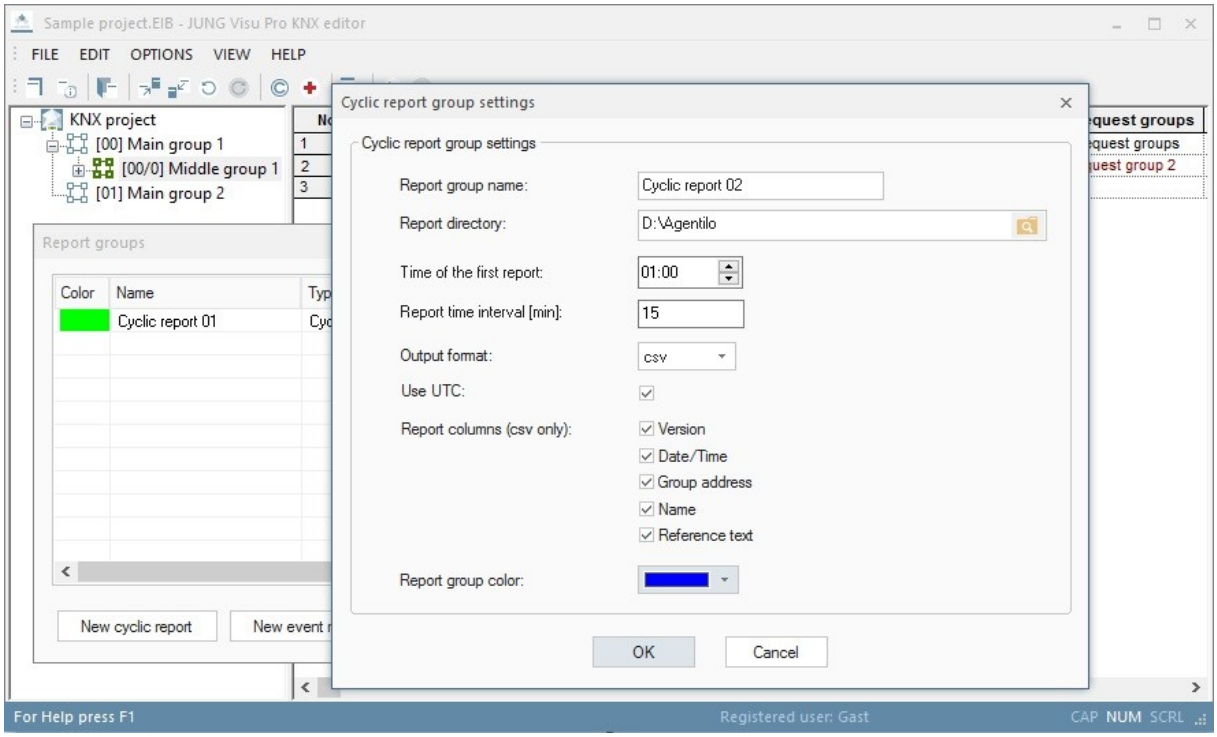
The report groups divide into: ***cyclic reports*** and ***event reports***.

Cyclic reports record from a certain time, in defined periods, the state of the data points within the group.

Event reports check after a preset time period if the value within the group changed and record this value in an event report. It can additionally be determined after what a time period of constant value the value should be recorded as an event.

Click under ***Options*** at ***Edit report groups*** to open the dialog ***Report groups***.

Create report groups:



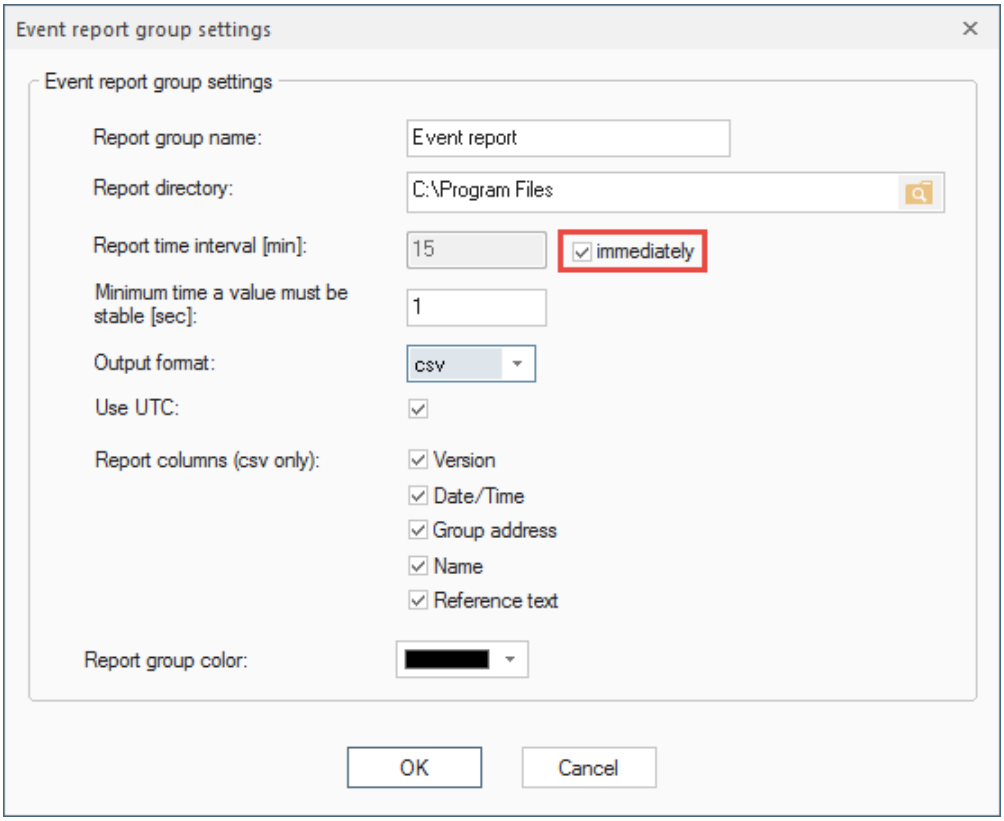
Click either at the button ***New cyclic report*** or ***New event report*** to open the dialog ***Report group – settings***. Enter the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Report groups***.

Report group name: Name of the group with which the group will be listed.

Report directory: The path to the folder that contains the saved reports.

Time of the first report (only at cyclic report): Time between 00:00 and 23:59.

Report time interval [min]: The time in minutes after which a new report is written. Oriented at the time of the first report.
Cyclic report: Must be between 1 and 1440 minutes, 1440 means a daily interval.
Event report: Must be between 0 and 1440 minutes, 1440 means a daily interval. If you set a check mark at ***immediately***, a report is created as soon as the event is detected. Alternatively you can enter a 0 at ***Report time interval***.



Minimum time a value must be stable [sec] (only at event report): Between 1 and 60 seconds.

Output format: The reports are written either into a xml or a csv file.

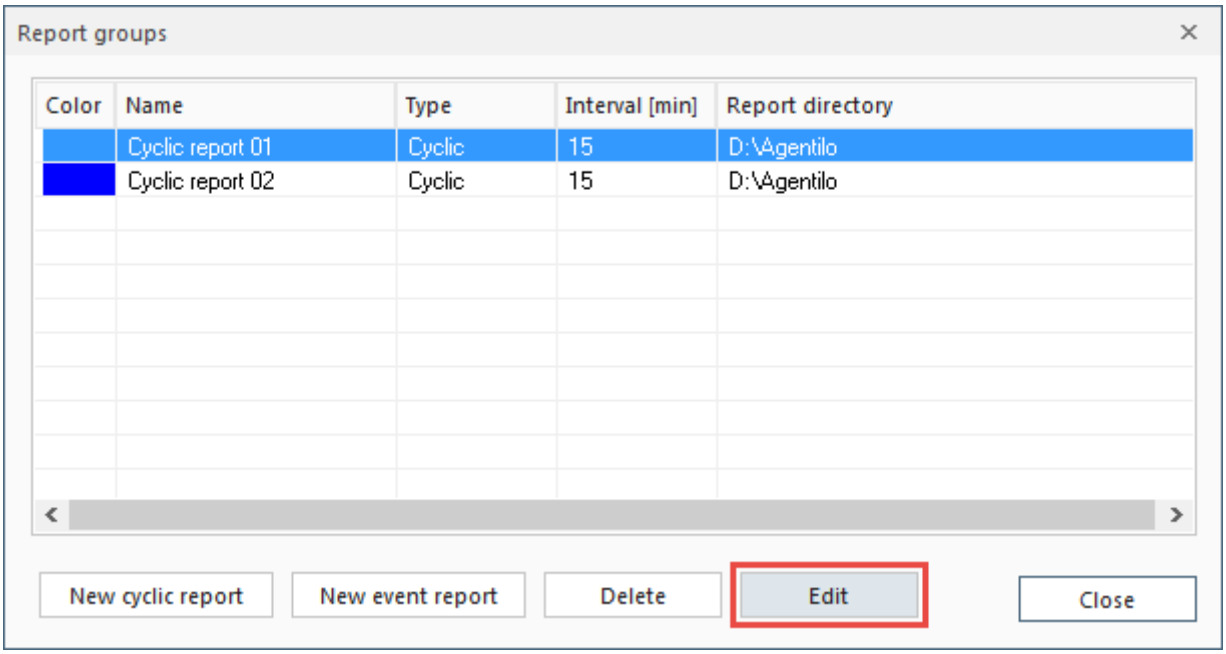
Use UTC: Activates or deactivates the Unit-Time-Code for writing the report.

Report columns (csv only): Defines which additional information should be written into the report.

Report group color: For a better discrimination of the different groups, a color can be set.

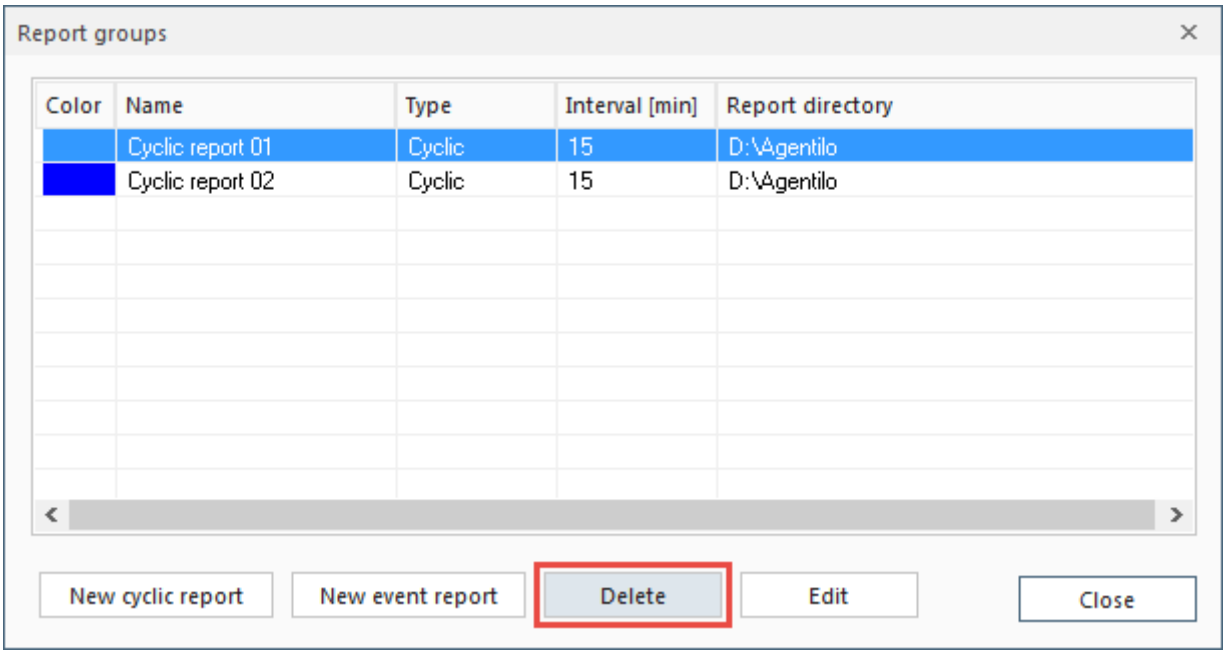
Edit report group:

To edit a report group, select the report group and click at ***Edit***. The dialog ***Report group settings*** opens.



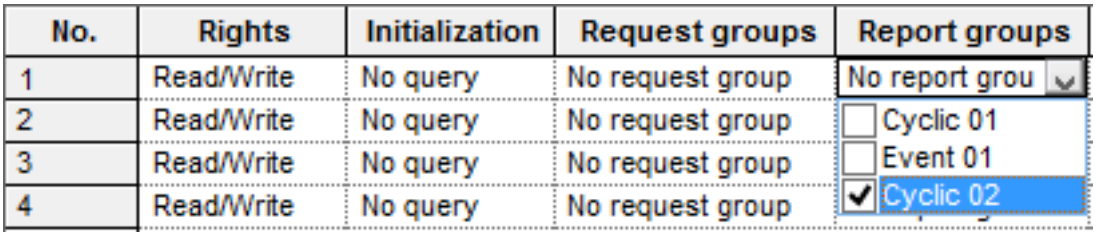
Edit the group settings and confirm the changes with ***OK***.

Delete report group:



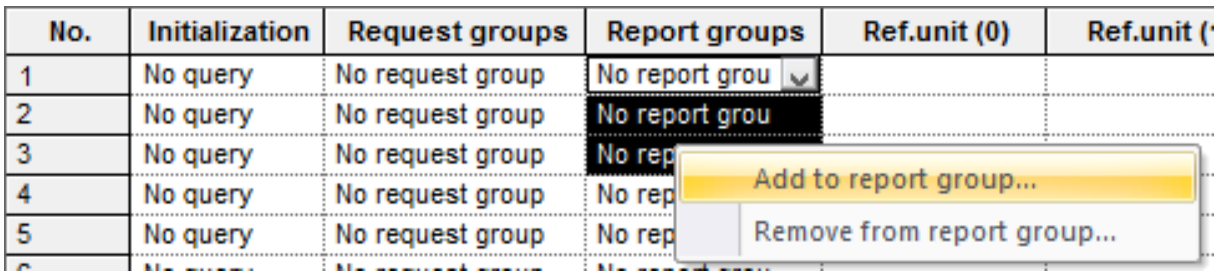
To delete a report group, select the report group and click at ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Report groups***.

Assign report group:

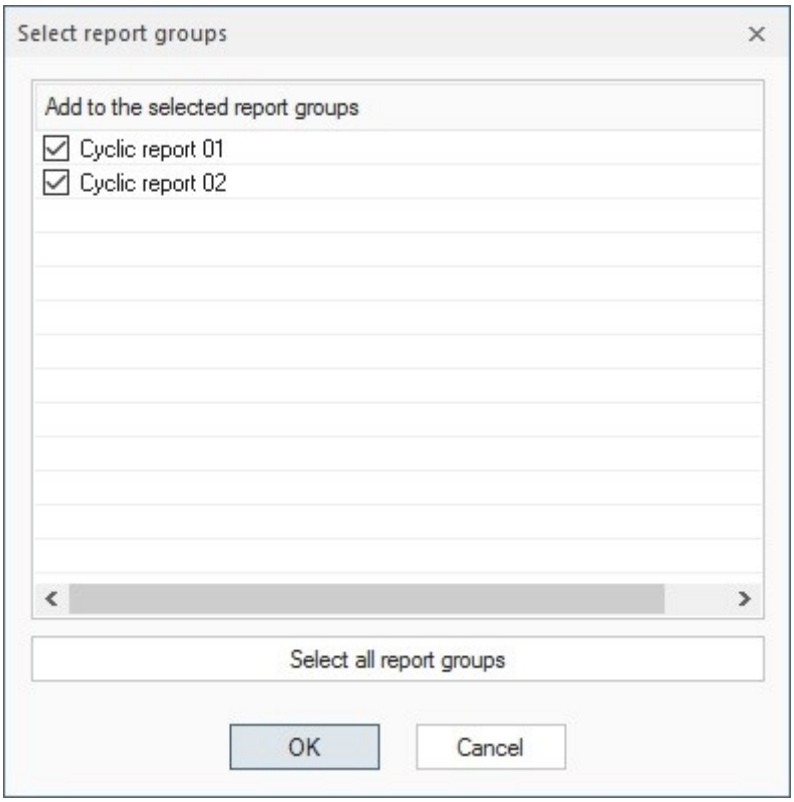


To assign a report group to a group address, click at the column ***Report groups*** to select a report group.

Or: To assign a selection of group addresses to a report group, select the first group address in the column ***Report groups*** and draw the mouse over the beneath lying lines. Right-click at the selected elements to open the context menu and click at ***Add to report group***.



The dialog ***Report group*** opens.



Set a checkmark at the desired report groups and confirm the selection with ***OK***.

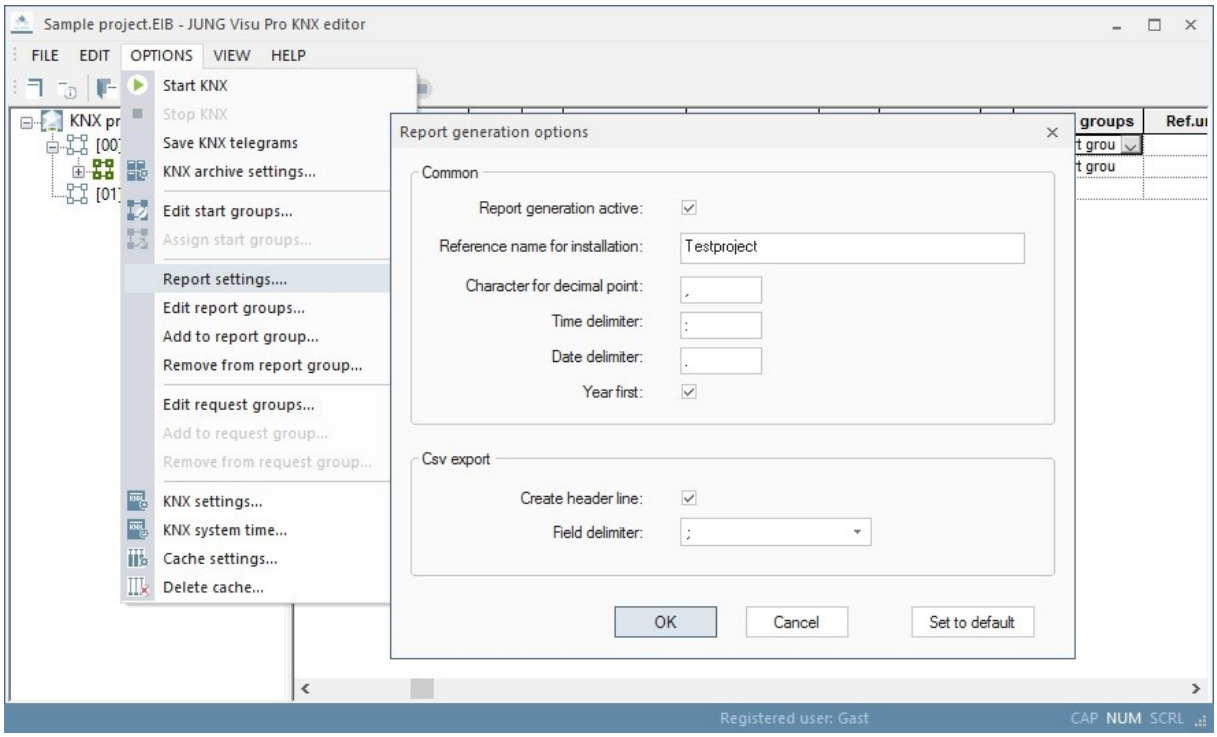
No.	Rights	Initialization	Request groups	Report groups
1	Read/Write	No query	No request group	3 report group
2	Read/Write	No query	No request group	3 report group
3	Read/Write	No query	No request group	Cyclic report 0
4	Read/Write	No query	No request group	Event report 01
5	Read/Write	No query	No request group	Cyclic report 0
6	Read/Write	No query	No request group	3 report group
7	Read/Write	No query	No request group	2 report group
8	Read/Write	No query	No request group	2 report group

For every group address is the report group now visible or, if a group address has multiple report groups, the number of report groups.

Delete report groups:
Deleting report groups is analog to assigning report groups.

4.6.1 Report Formatting

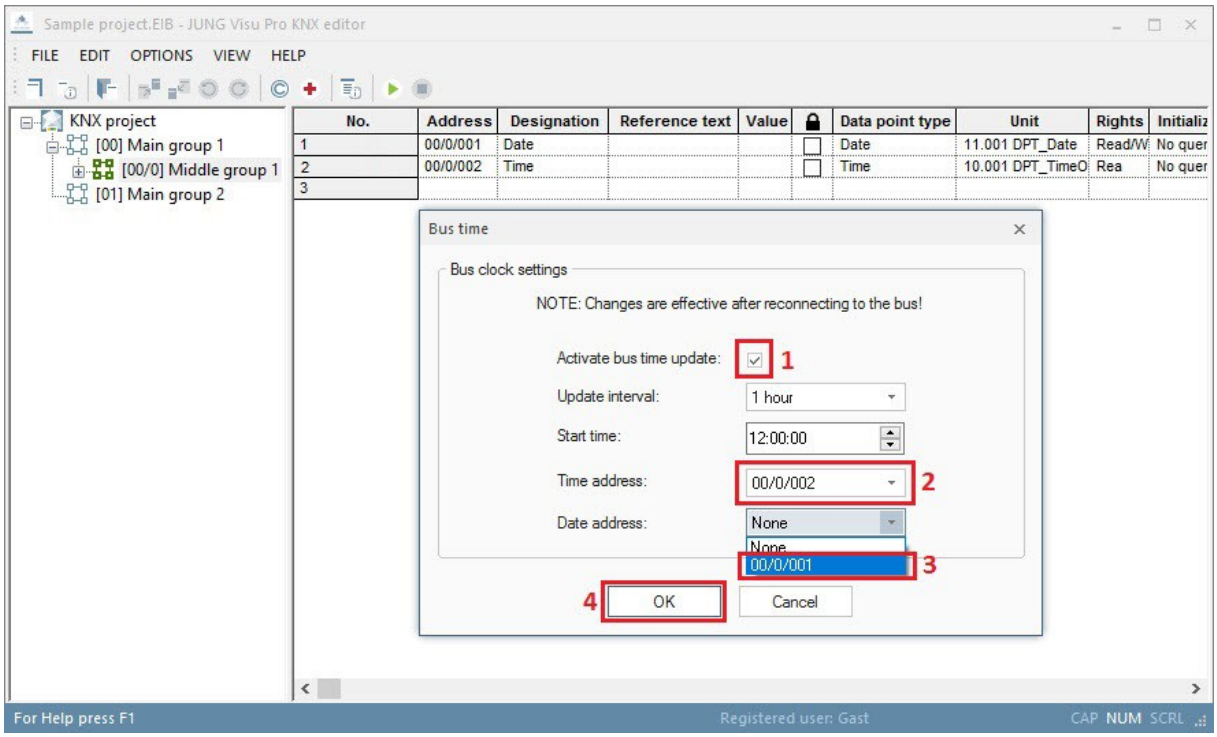
If the produced csv format is not of use for further editing, it is possible to change the formatting of the reports. Click under ***Options*** at ***Report settings*** to open the dialog ***Options for the report creation***.



Change the settings and confirm with ***OK***.

4.7 KNX System Time

To activate the actualization of the bus time, choose the menu item ***Options*** and click ***KNX system time***. The dialog ***Bus time*** opens.



To **actualize** the bus time **automatically**, activate the check box ***Activate bus time update***.

Set the update interval and the start time and choose the write addresses for date and time. Press ***OK*** to commit the changes and to close the dialog.

4.8 Synchronize PC Time with KNX Time

By synchronizing bus and PC time, the difference between bus and PC time shall be minimized.

The synchronization mechanism is not a method for setting a specific time or date on a PC via bus telegrams. In particular, it is also not intended to perform the summer/winter time changeover on a PC.

The synchronization is only possible within a certain time slot. The time information from the bus is evaluated by the KNX editor between 5:00 and 22:00. The synchronization will be done if the difference between bus and PC time is greater than or equal to 5 seconds.

To do so the KNX editor evaluates 2 bus telegrams: One telegram containing the current date (DPT 11.001) and one telegram containing the current time (DPT 10.001). The KNX editor takes the bus time into account only if the interval between these two telegrams is less than 10 seconds.

A suitable KNX device (e.g. weather station) must be used and configured for the cyclical updating of the bus time.
Possible update intervals are:

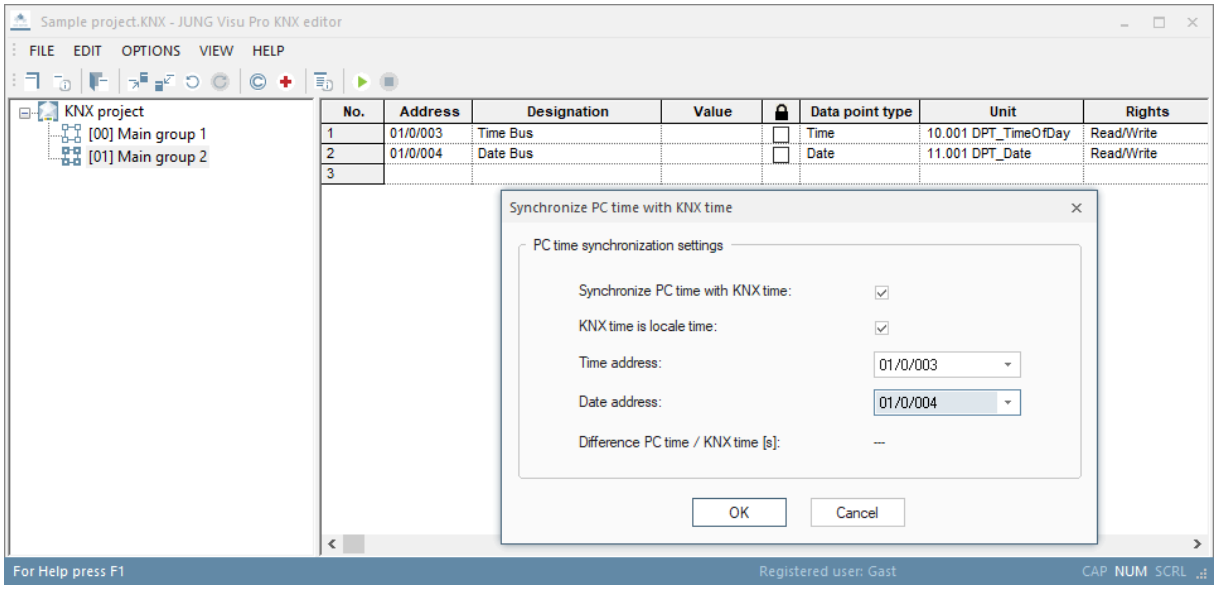
- Hourly transmission of the bus time
- Daily transmission of the bus time.
  If the bus time is transmitted once a day, the transmission time must be within the time slot described above, e.g. at 6:00 a.m.

Depending on the KNX device, the transmission of the current time can also be triggered via a group address. In this case, a corresponding Request Group can be created in the KNX editor, which queries the bus time every hour or in 24 hour intervals.

Recommendation:

- Daily sending of the time and date between 05:00 and 22:00.

To synchronize the PC time with the KNX time, click on ***Options*** in the menu and select ***Synchronize PC time with KNX time***… Following dialog will be opened:



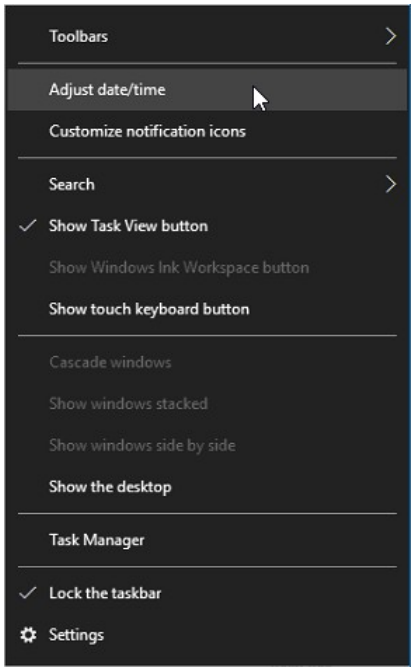
To **automatically synchronize** the PC time you have to enable the checkbox ***Synchronize PC time with KNX time***.

Choose the read addresses for date and time.

If the KNX time does not match the locale time zone, but UTC instead, you have to disable the checkbox ***KNX time is locale time***.

Press ***OK*** to commit the changes and to close the dialog.

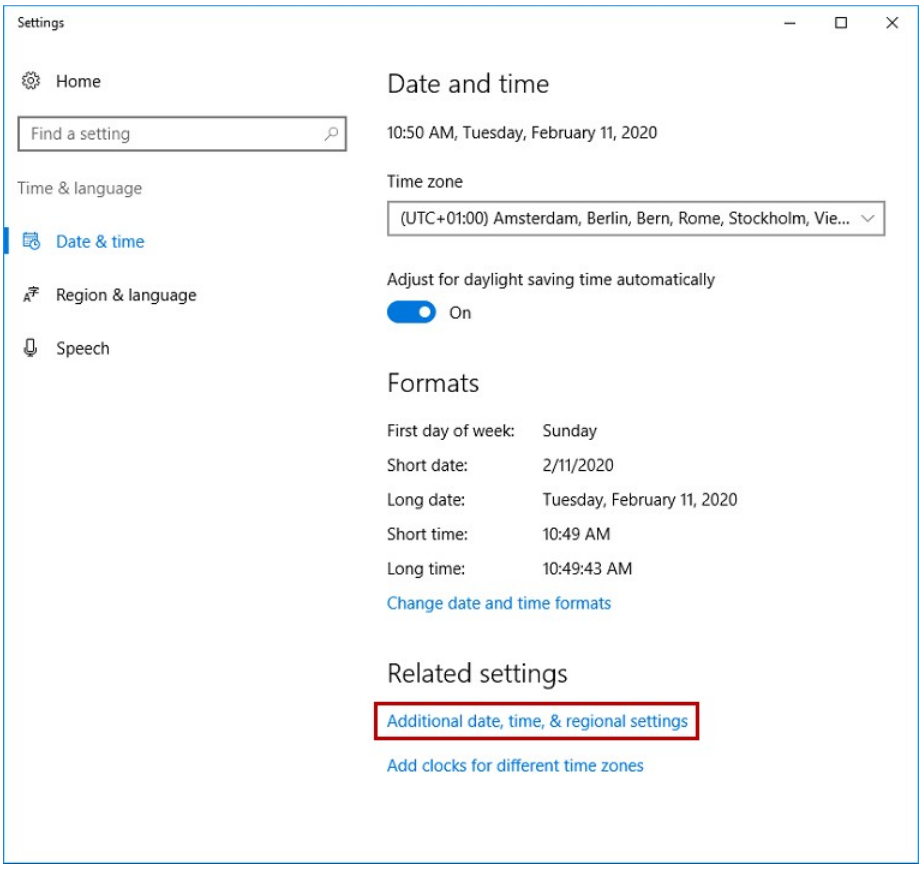
To prevent Windows 10 from constantly updating the time, Internet synchronization must be disabled in the time settings:



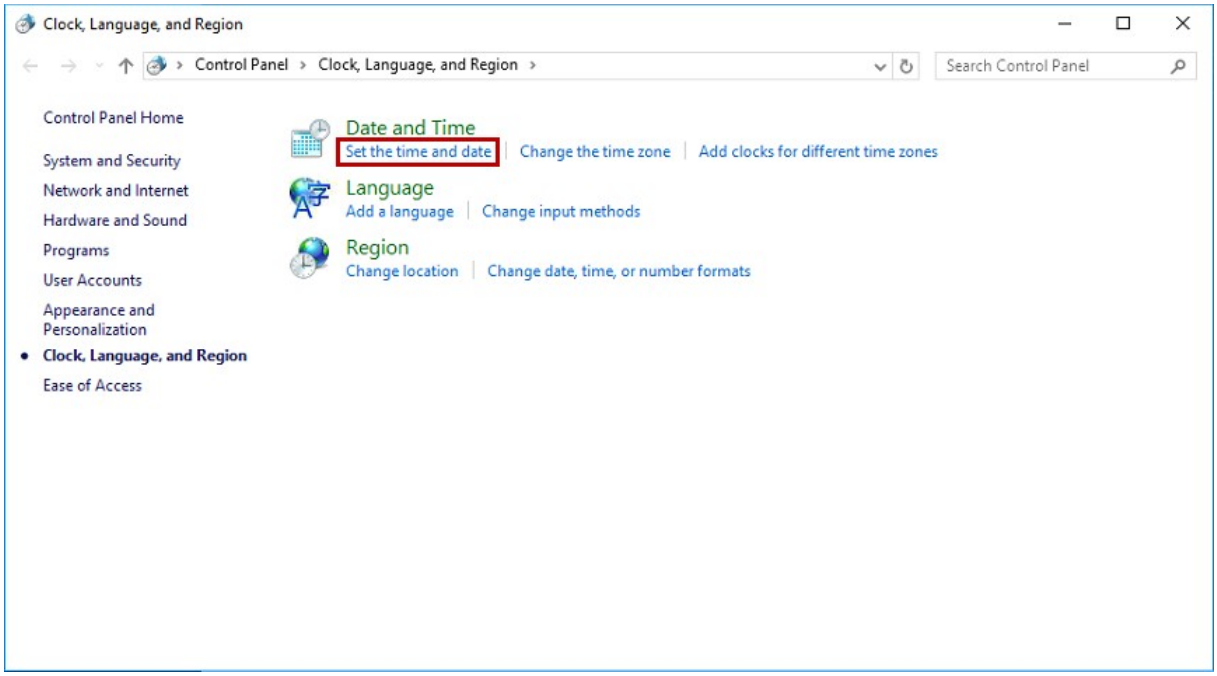
click on the clock with the right mouse button (at bottom on the right on the screen)

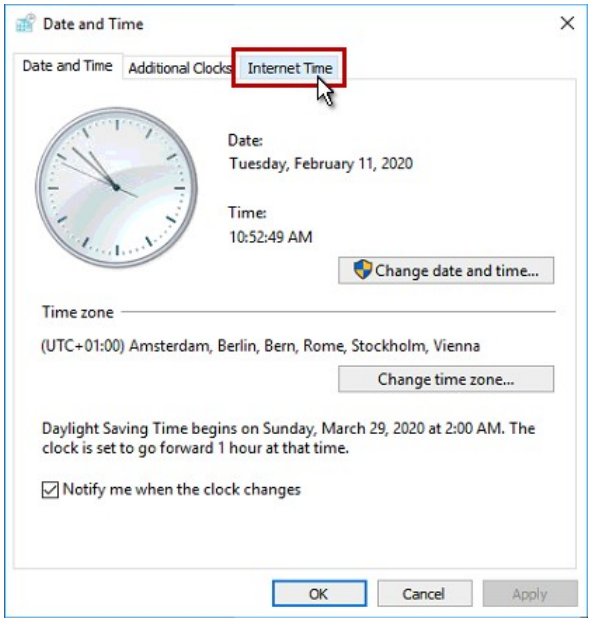
select ***Adjust date/time***

under *Related settings*, click on ***Additional date, time, & regional settings***

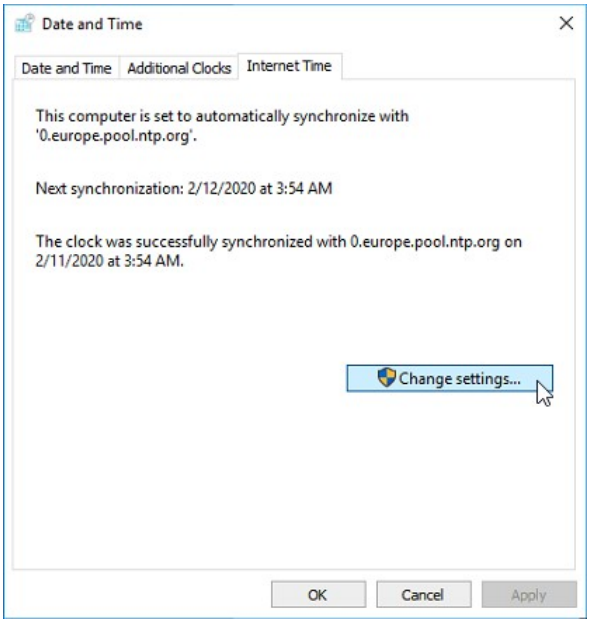


then click on ***Set the time and date***

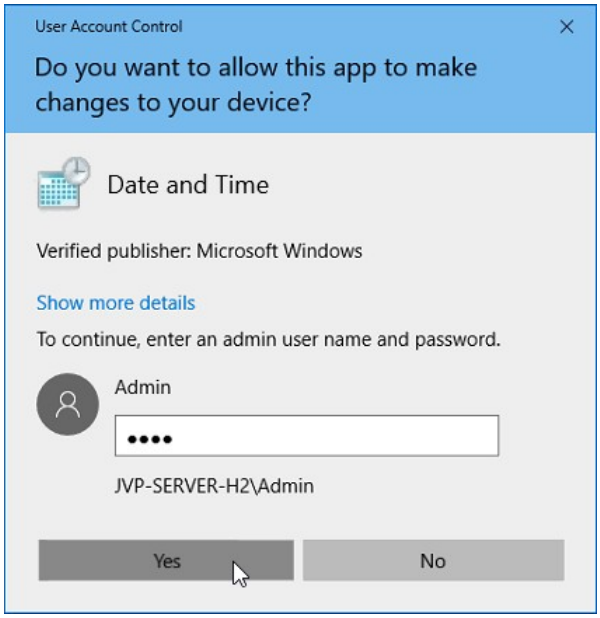




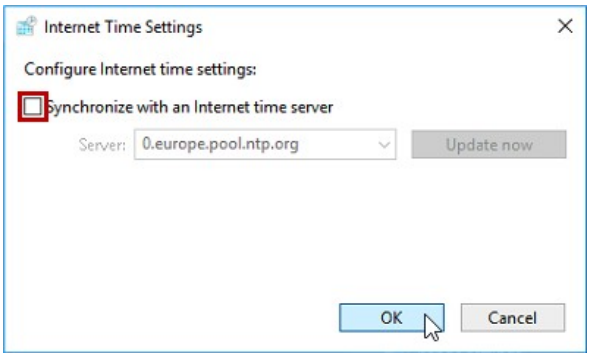
then select ***Internet Time***



click on ***Change settings...***



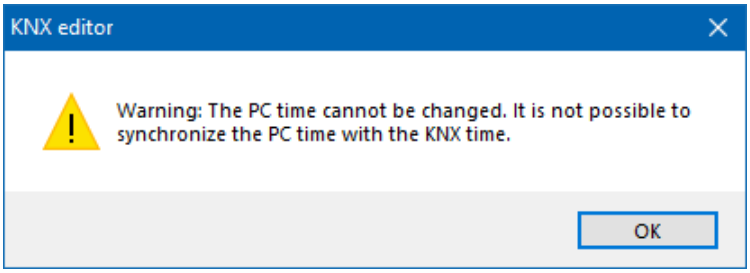
Enter the password of the administrator, click ***Yes***



Remove the check mark for ***Synchronize with an Internet time server***, confirm with ***OK***

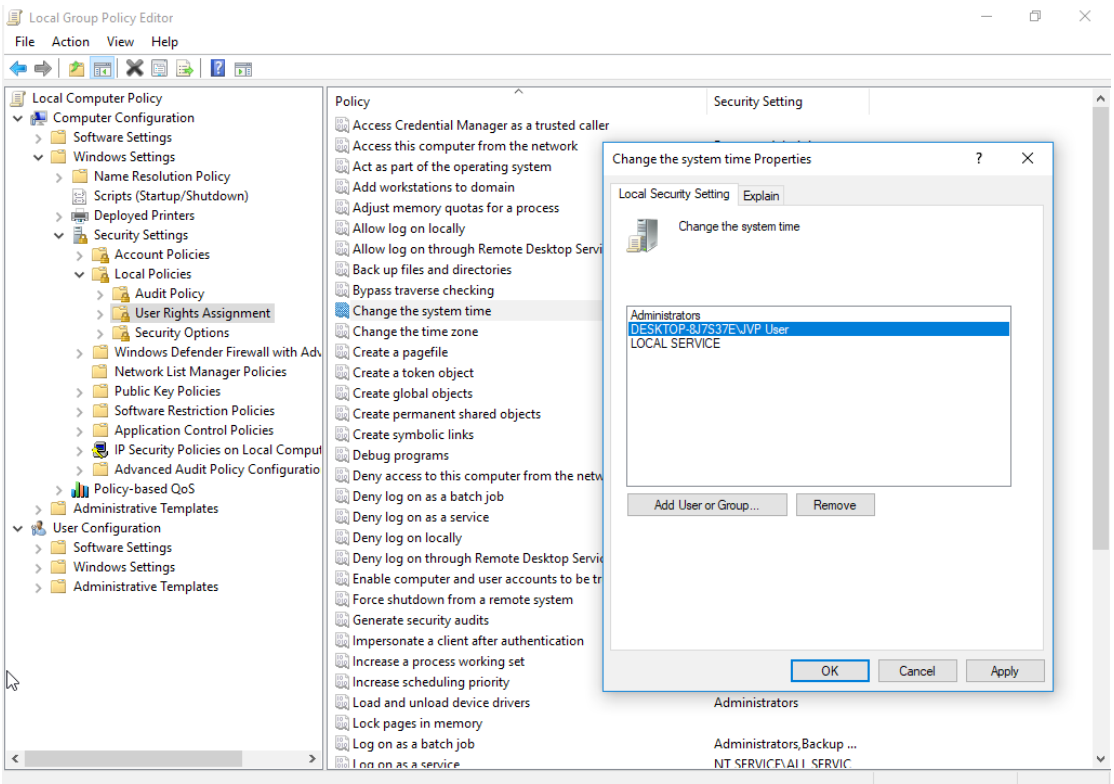
Warning:

To change the PC time, the KNX-Editor needs Administrator rights! A dialog will be shown, if you click ***Synchronize PC time with KNX time*** *and do not have the needed rights.*



To open the KNX-Editor with administrator rights, right click on JUNG VISU Pro and choose ***Run as administrator****.*

Alternatively, you can grant your normal Windows user the necessary rights to change the PC time. This is necessary if the editor starts automatically.
To do this you need to configure the local group policies, which are ***not available*** *for home editions of Windows.*
To make changes in the 'Local Group Policy Editor', ***it is necessary to change the user and log in as 'Admin'.*** *From the list choose Computer Configuration → Windows Settings → Security settings → Local Policies → User Rights Assignment and choose* ***Change the system time****. A dialog opens, which allows you to add your current user for this policy. Restart your system after you added the user.*



4.9 Application of KNX Data Types

The KNX data point types specify the meaning of transferred bus data. There are different interpretations of the raw data depending on their data type.

The data packet **cf c7**, received by the KNX editor, can have completely different meanings:

-291.84,	interpreted as 2 Octet Float Value
53191,	interpreted as 2 Octet Unsigned Value
–12345,	interpreted as 2 Octet Signed Value

Attention: *The correct assignment of KNX data point types to group addresses is a fundamental precondition for all applications which use the KNX editor as process connection!*

The following table gives a short introduction to the meaning of KNX data point types:

Boolean	**Unit** **1.001 –1.022** **DPT**	**1 bit** data length, formerly EIS1. This data type is used to transmit switch states (e.g. 0/1, on/off, open/closed, up/down, …). This type is particularly used to switch devices on or off or to receive device states (lamps, dimmers, shutters, relay, …).
1-Bit Controlled	**Unit** **2.001 – 2.012** **DPT**	**2 bit** data length, formerly EIS8. This data type is used for priority-controlled switch operations.
3-Bit Controlled	**Unit** **3.007 – 3.008** **DPT**	**4 bit** data length, formerly EIS2, subtype control. This data type is used to increase or decrease the set value in steps or stop the movement (e.g. to control a dimmer).
Character Set	**Unit** **4.001- 4.002** **DPT**	**1 byte** data length, formerly EIS13. This data type is used to transfer a character over the bus. ***Attention:*** *The visualization does not yet support this data type.*
8 Bit Unsigned Value	**Unit** **5.001, 5.003, 5.004, 5.010**	**1 byte** data length, formerly EIS6 or EIS14. This data type is used to transfer 8 bit unsigned integer values (e.g. counter values).

Attention: *Normally the range of the 8 bit value is from 0 to 255, but the interpretation can differ depending on the subtype selection. If the subtype* scaling *is selected, the value is interpreted as a percent value in the range from 0 to 100%. This subtype can be used to set the values of dimming actuators. If the subtype* wind direction *is selected, the value is interpreted as an angle in the range from 0 to 360°.*

Subtype 5.001 DPT_scaling	The value is interpreted as a percent value in the range from 0 … 100%.
Subtype 5.003 DPT_angle	The value is interpreted as an angle in the range from 0 … 360°.
Subtype 5.010 DPT_Value_Ucount	The value is interpreted in the range from 0 … 255, e.g. as counter value.

8 Bit Signed Value	**Unit 6.010 DPT**	**1 byte** data length, formerly EIS14, used to transfer an 8 bit signed integer value in the range from –128 to +127.
2 Octet Unsigned Value	**Unit 7.001, 7.010, 7.600 DPT**	**2 byte** data length, formerly EIS10, used to transfer a 2 byte unsigned integer value (e.g. a counter value).
2 Octet Signed Value	**Unit 8.001 DPT**	**2 byte** data length, formerly EIS10 signed, used to transfer a 2 byte signed integer value in the range from – 32768 to +32767.
2 Octet Float Value	**Unit 9.001-9.006, 9.010-9.011. 9.020-9.021 DPT**	**2 byte** data length, formerly EIS5, used to transfer a 2 byte analog value (e.g. values from a temperature sensor).
Time	**Unit 10.001 DPT**	**3 byte** data length, formerly EIS3, used to transfer a time information (e.g. from a KNX synchronization clock).
Date	**Unit 11.001 DPT**	**3 byte** data length, formerly EIS4, used to transfer a date value.
4 Octet Unsigned Value	**Unit 12.001**	**4 byte** data length, formerly EIS11, used to transfer a 4 byte unsigned integer value (e.g. a counter value).

4 Octet Signed Value	**Unit 13.001**	**4 byte** data length, formerly EIS11 signed, used to transfer a 4 byte signed integer value in the range from -2147483648 to +2147483647.
4 Octet Float Value	**Unit 14.001-14.079 DPT**	**4 byte** data length, formerly EIS9, used to transfer a 4 byte high precision analog value according to IEEE754 standard.
Access	**4 byte** data length, formerly EIS12, used for access control applications.	

> ***Attention:***
> *The visualization does not yet support this data type.*

String	**Unit 16.000-16.001 DPT**	**14 byte** data length, formerly EIS15, used to transfer strings.
UTF-8 String	**Unit 28.001 DPT**	**14 byte** data length, used to transfer strings in UTF-8 format.
Colour RGB	**Unit 232.600 DPT**	**3 byte** data length, used to transfer colors in RGB format.
DateTime	**Unit 19.100 DPT**	**8 byte** data length, used to transfer date and time.
Colour RGBW	**Unit 251.600 DPT**	**6 byte** data length, used to transfer colors in RGBW format.

4.10 ETS Import of KNX Group Addresses

In most cases it is more convenient to import KNX group addresses from ETS than to create them manually. The KNX editor is able to import KNX group addresses from **ETS2** versions 1.1, 1.2 and 1.3.

There are two methods:

a) Import the data from an ETS print report which has been redirected to a file. This method applies to **ETS2**, versions 1.1, 1.2 and 1.3.
b) Import the data from a file which has been created with the function OPC export. This method applies to **ETS2**, Version 1.3 and **ETS3**.

Data from **ETS4** and **ETS5** can be exported to a **project export** file which can be imported in the KNX editor.

> ***Important:*** *The KNX editor does not require ETS to be installed at the same PC. It is sufficient to copy the ETS output files to the PC.*

Please consider the following notes when importing group addresses from ETS:

- Assign **unique names** to the group addresses in the ETS project. It is important to assign meaningful names to the group addresses in order to facilitate the identification of addresses and plane text messages in the KNX editor and the visualization.

- Make sure to set the **data point types** already in the ETS project. The KNX editor is able to import the data point types from the ETS print file. If you have configured the data point types correctly in the ETS project, you do not have to enter them again in the KNX editor. This facilitates in particular the updated import of addresses after changes have been carried out in the ETS project.

- Edit the imported group addresses after the import to the KNX editor:

  → Check whether the **data point types** are set correctly;

  → Set the **read and write rights**. The read and write rights for the KNX editor (and for programs using the KNX editor) **cannot** be imported from ETS. They must be set manually.

4.10.1 Create ETS Group Addresses Print Report

Create ETS export file in ETS:

1. Open the ETS project in ***Project Design***.
2. Open the group address browser (Menu: ***Design***, ***Group Addresses***).
3. Choose the menu item ***Project*** > ***Page Setup*** and click ***Layout***. Click in the layout dialog ***reset the selection of Title page, Header, Footer***.
4. Mark the **root node** in the group address browser.
5. Choose the menu item ***Project*** and click ***Print***. In the print dialog click ***Print to File*** and click the button ***OK*** to start the export.
6. A print dialog opens. Choose the target directory, target file name, and select the extension ***.txt***.
7. To import the output file into the KNX editor, choose the menu item ***File*** and click ***Import ETS project***.

> ***Important:*** *With ETS the installed printer driver must support the function* **printing in a file***. As a remedy install the WINDOWS standard printer driver.*

4.10.2 Import OPC Export File from the ETS

The KNX editor can import group addresses from a file which has been created from ETS2 version 1.3, ETS 3 and ETS4 via the function ***OPC Export***.

Create OPC export file in ETS2 version 1.3:

1. Start the function ***OPC Export*** in ETS.
2. The ***OPC Export*** dialog opens. Select the ETS project and specify a target directory and a file name for the OPC project file.
3. Click the button ***Export*** to create the file in the specified directory. The default extension of the OPC project file is ***.esf***.
4. To import the file in the KNX editor, choose the menu item ***File*** and click ***Import ETS project***.

Create OPC export file in ETS3:

1. Choose the menu item ***Extras*** and click ***Options***. Select the tab ***Presentation***. Select under ***Browser*** the check box ***Two Level Group Addresses***.

2. Choose the menu item ***File*** and click ***Extract Data (e.g. OPC)***. The dialog ***Foreign Format*** opens.

3. Click ***Export to OPC Server***. Specify the target directory and a file name for the OPC project file. Then click ***Export***. The default extension of the OPC project file is ***.esf***.

4. To import the file into the KNX editor, choose the menu item ***File*** and click ***Import ETS project***.

5. To toggle between the 2-level, 3-level and the free address structure, choose the menu item ***Edit*** and click the desired address structure.

Create OPC export file in ETS4 and ETS5:

1. Switch in ETS4 to the project overview and select the desired project. The button ***Export*** in the toolbar will be activated.

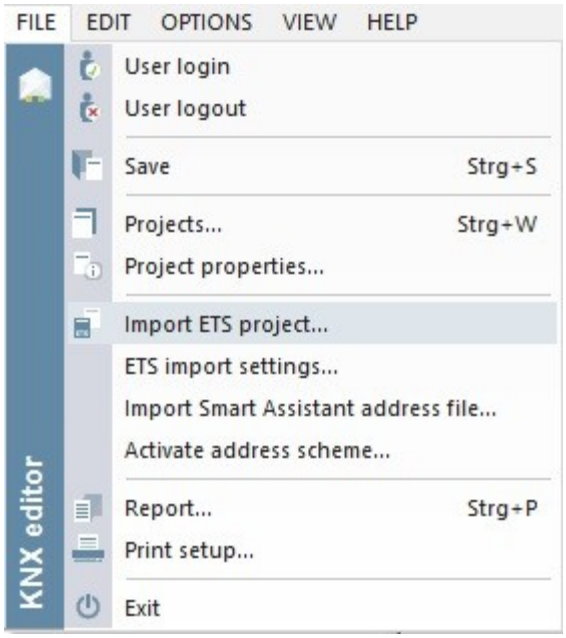
2. Click in the toolbar the button ***Export***.

3. Enter the project name *"Projectname.knxproj"* and click the button ***Save***. The default extension of the file is ***.knxproj***.

4. A dialog indicating the progress of the data transfer opens. After the transfer is completed, confirm the pop up message box ***Operation successful*** by clicking the button ***OK***.

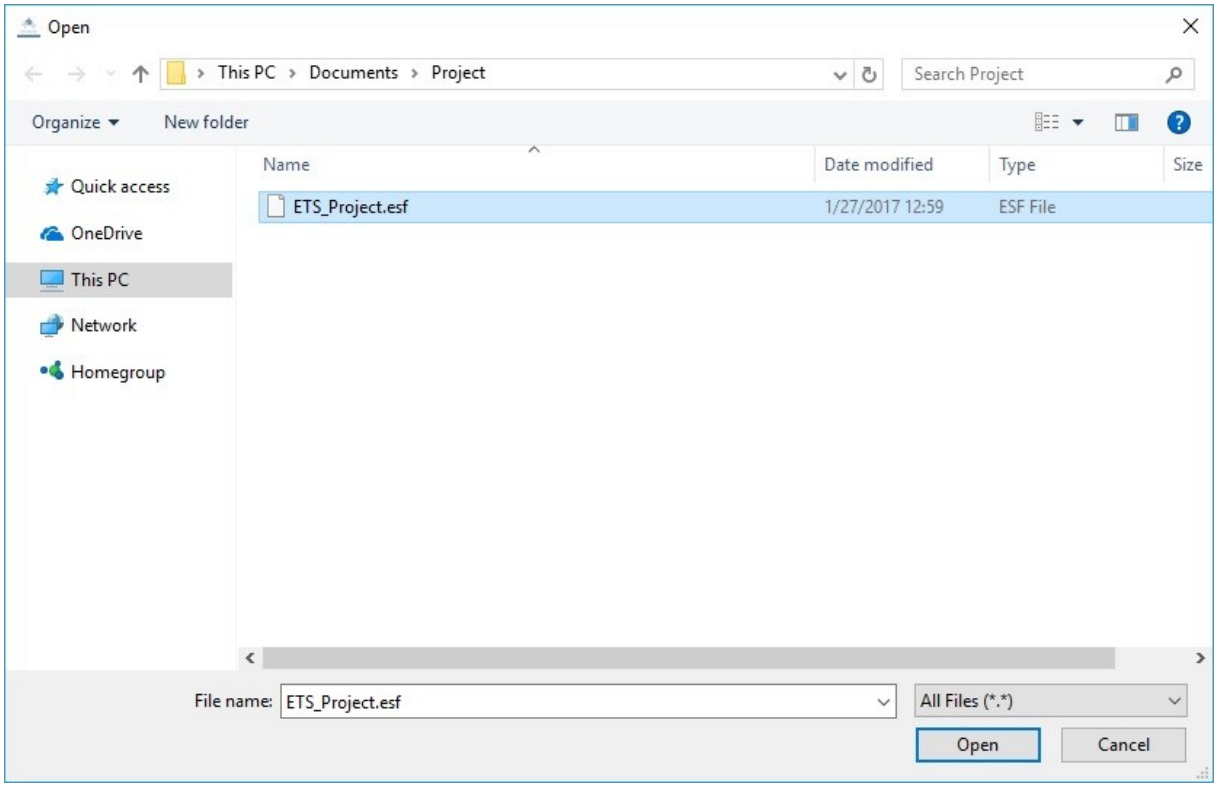
5. The exported ETS project is now available in the selected directory and can be imported in the KNX editor.

4.10.3 Import of ETS Export Files into the KNX Editor



Import ETS project:

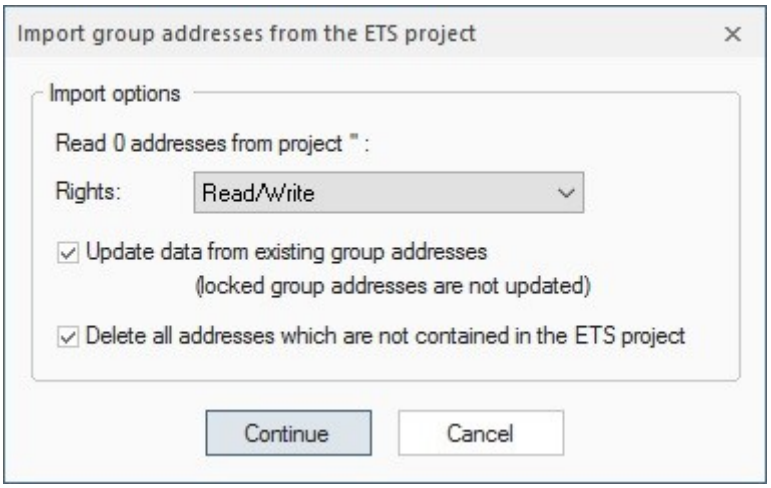
Choose the menu item ***File*** and click ***Import ETS project*** to open the dialog for the selection of the directory.



The redirect print reports usually have the extension ***.txt***, the OPC Export-generated files have the extension ***.esf*** and the ETS4 export files have the extension ***.knxproj***.

Select the file to be imported and click the button ***Open***. A dialog for the import opens. Select the project out of the list from the ETS file and confirm with ***OK***.

A dialog for the import of the group addresses opens.



The dialog shows the number of group addresses in the import file. You can set here the access rights which will be assigned to the imported group addresses by default. Since the rights for read and write access cannot be imported, make sure to check them and, if necessary, correct them after the export. But at this step you may select the default rights which are most convenient for the project.

Press ***Continue*** to load the ETS project into the editor.

Import options:

With the check boxes at the import dialog, the following options can be set:

Update data from existing group addresses:

If this option is set, the data of already existing group addresses in the KNX editor will be actualized.

Note: *Enable this option if you want to transfer the changes carried out in the ETS project to the group addresses already contained in the KNX editor (i.e. change of address names).*

Lock and unlock group addresses:

You may want to prevent a data point from being updated. To **lock a group address** for the update, select the checkbox in the column with the padlock icon.

No.	Address	Designation	Value	🔒	Data point type	Unit	Ri
1	00/0/006	Group 6		☑	Boolean	1.001 DPT_Switch	Read/W
2	00/0/007	Group 7		☐	Boolean	1.001 DPT_Switch	Read

To lock several group addresses, click a group address in the column with the padlock icon, hold down the Ctrl key and extend the selection on further group addresses. Then right-click and click in the context menu the option ***Lock address***.

To lock a range of group addresses, click the first group address in the column with the padlock icon, hold down the shift key and click the last group address to be locked. Alternatively, click with the left mouse button the first address to be locked and move the mouse cursor to the last address to be locked. Then right-click and click in the context menu the option ***Lock address***.

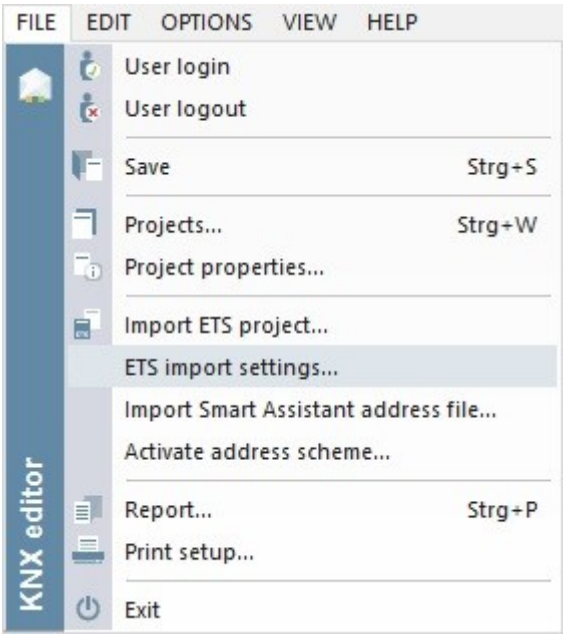
To unlock an address for the update, clear the check box or right-click and click in the context menu the option ***Unlock address***.

> ***Note:*** *If you check the option* ***Delete all addresses which are not contained in the ETS project****, the locked group addresses will also be deleted if they are not contained in the ETS project.*

Delete all addresses, which are not contained in the ETS project:

If this option is enabled, all addresses will be deleted which are available in the KNX editor but not in the ETS project.

4.10.4 ETS-Import Settings...

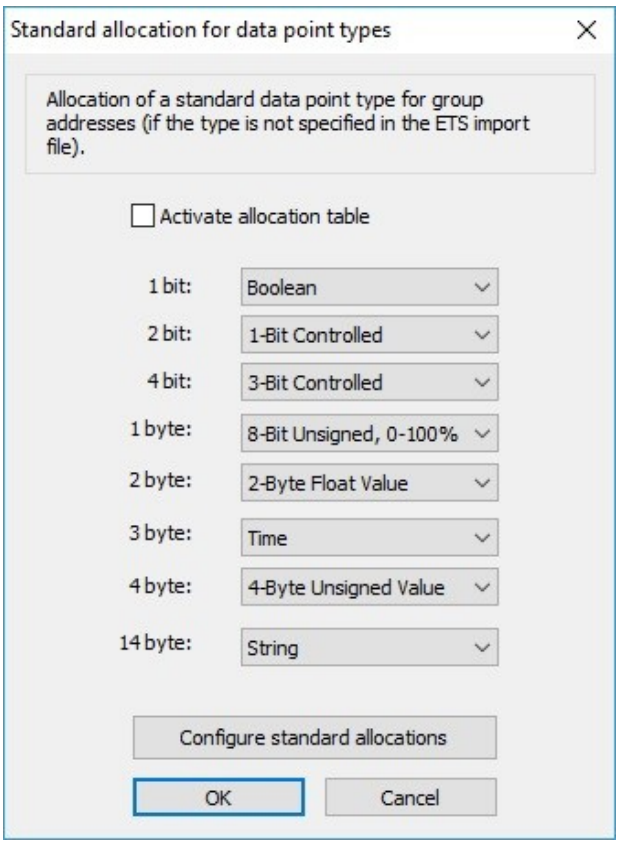


ETS-Import Settings:

Choose the menu item ***File*** and click ***ETS-Import Settings...*** to open the dialog for the standard type declaration for data points.

In this dialog it is possible to activate the declaration table. The decalration table sets standard types for every data point, which has no type specified after the importation of an ETS file.

You can specify a standard data point type foreach available data point length.



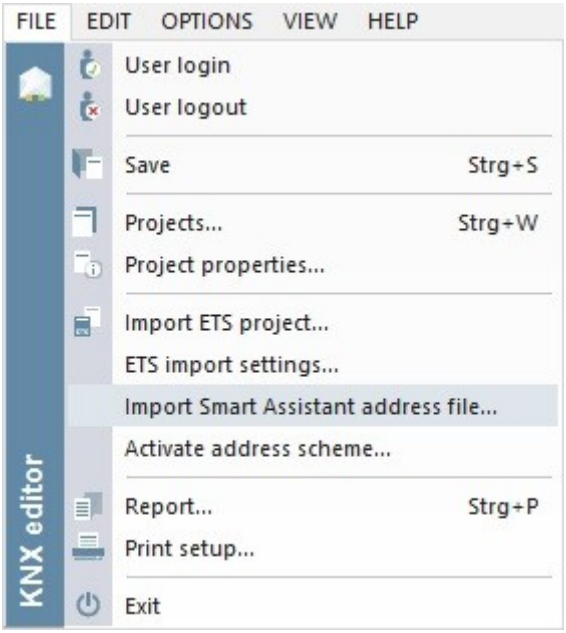
Following data point types are the default values of the declaration table:

1 Bit	Boolean
2 Bit	1-Bit Controlled
4 Bit	3-Bit Controlled
1 Byte	8-Bit Unsigned, 0-100%
2 Byte	2-Byte Float Value
3 Byte	Time
4 Byte	4-Byte Unsigned Value
14 Byte	String

These standard declarations can be restored by clicking the ***Reset to standard declarations*** button.

4.11 Import of Smart Assistant Address Files

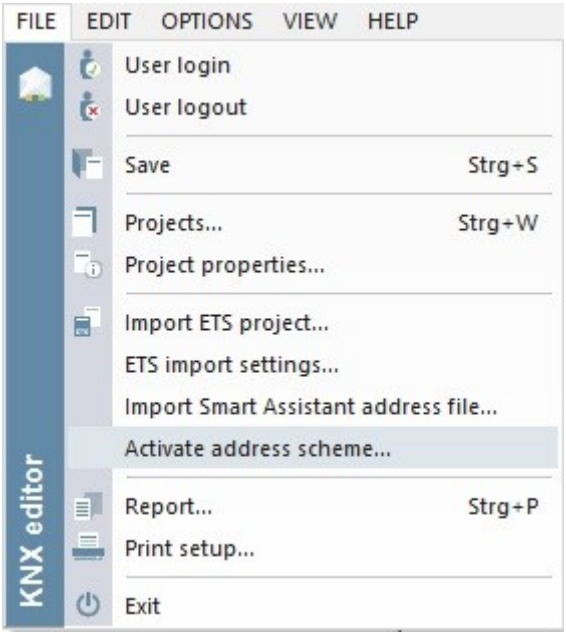
It is possible to import address files, generated by the JUNG Visu Pro Smart Assistant, into the KNX editor.



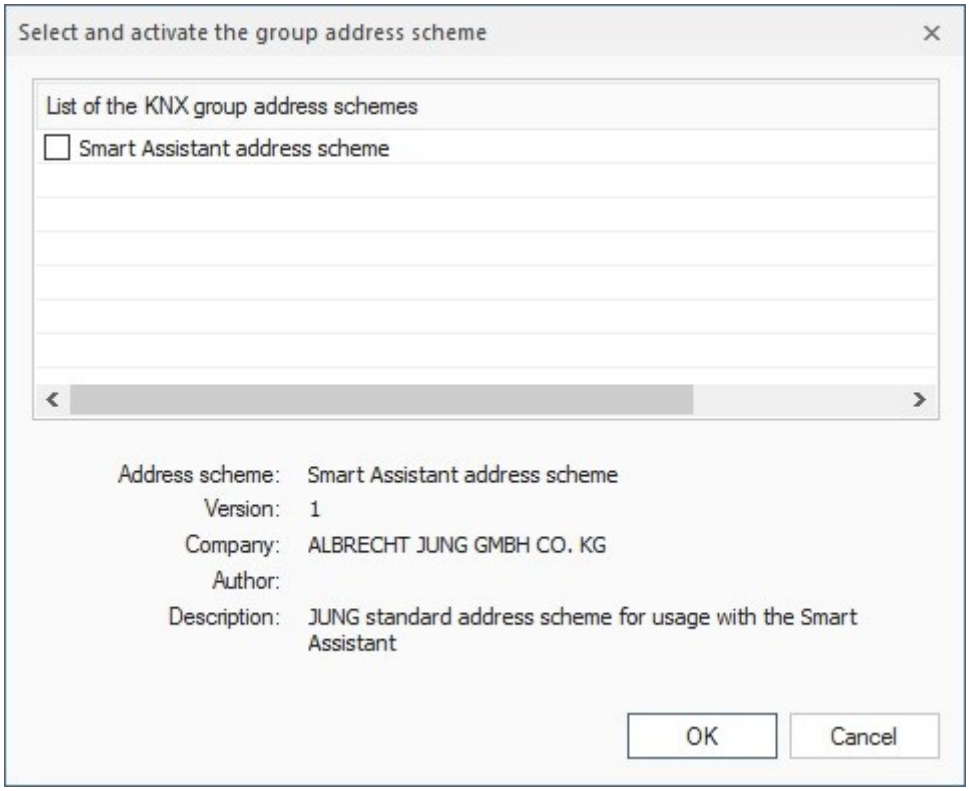
Import of a Smart Assistant Address File:

Choose the menu item ***File*** and click ***Import of a Smart Assistant Address File...*** to open the dialog for the selection of the address file.

4.12 Activate Address Scheme

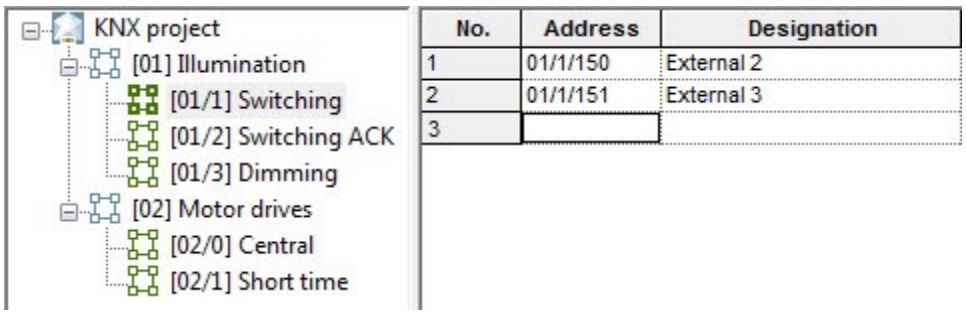


Choose the menu item ***File*** and click ***Activate Address scheme...*** to open the dialog for the selection of the address scheme.



This dialog offers schemes, which can be activated to define specific labels for main and middle groups or to set a standard datapoint type for datapoints in specific middle groups. At the moment the only available scheme is the JUNG Visu Pro Smart Assistant scheme.

The JUNG Visu Pro Smart Assistant scheme offers the same structure for your KNX project, which is used when creating a Smart Assistant project. There is no need to define the labels on your own.



4.13 KNX Settings

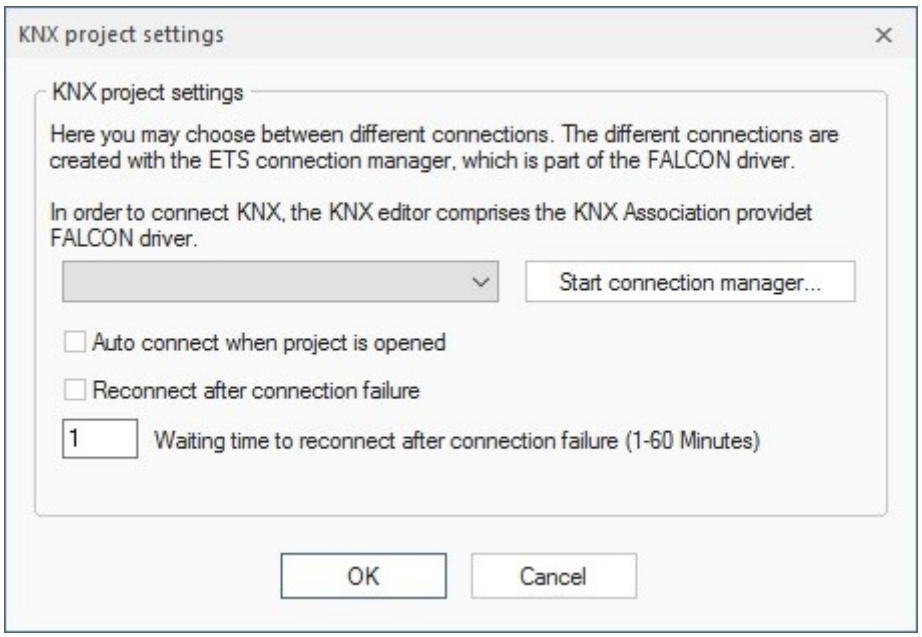
Connect PC to KNX:

The connection to KNX is realized via the FALCON driver (of the KNX Association) which is contained in the KNX editor. The Falcon driver supports the following interfaces: RS 232 (PEI10 and PEI16), KNX-USB, EIBlib/IP and KNXnet/IP. The PC must provide one of the following interfaces: RS232, USB, IP.

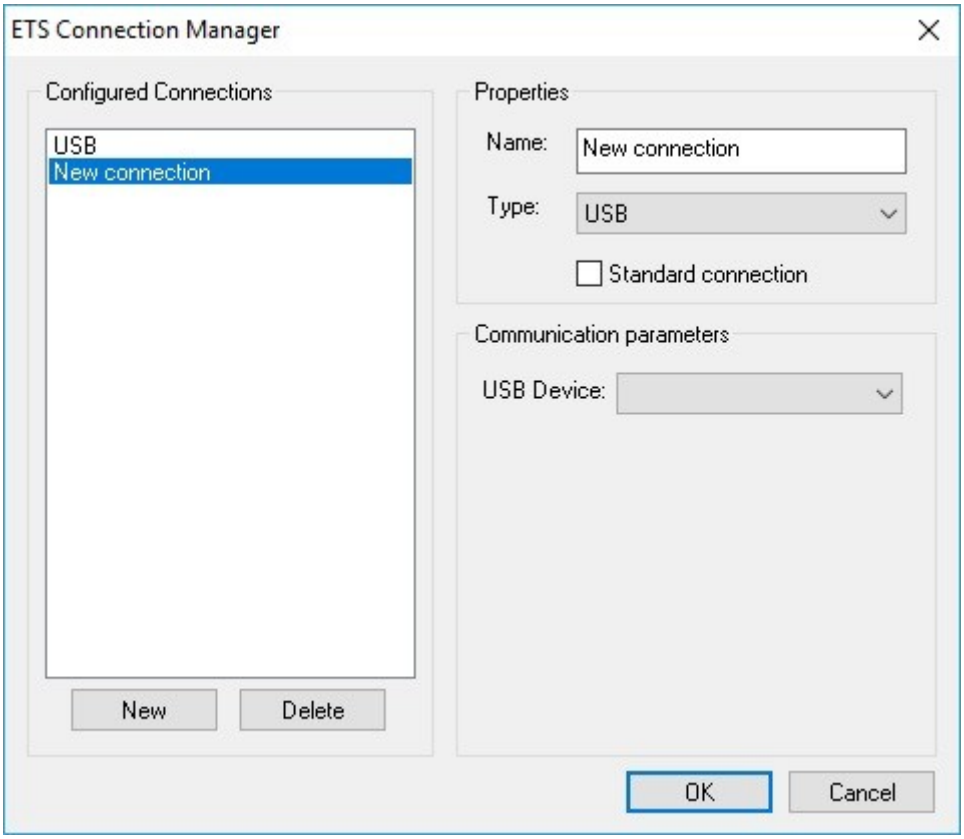
Set KNX parameters:

In the KNX editor the Falcon driver is configured in such a way that the connection can be established via one of the interfaces. The connection to the serial interface of the BCU is possible via the handshake protocol (PEI16) or FT1.2 protocol (PEI10).

Choose the menu item ***Options*** and click ***KNX settings***. The dialog ***KNX project settings*** opens.

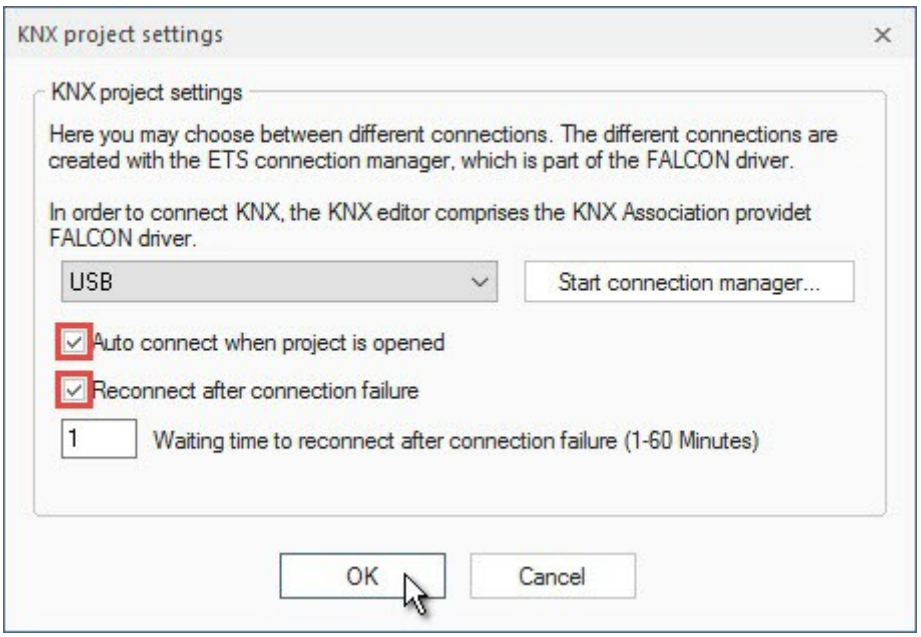


Here you can specify which of the KNX connections configured for the PC should be used to establish the access to the bus. The different connections are created with the help of the ETS connection manager which is part of the FALCON driver. Click the button ***Start connection manager*** to start the ***ETS connection manager*** dialog.



Click the button ***New*** to create a new connection. In the example below a standard connection for the USB interface has been created. The name can be edited. The connection type and the USB device can be selected.

Click ***OK*** to commit the entries.



Auto connect when project is opened:

The connection to the bus can be established automatically right after the start of the KNX project. To enable this option, select the check box ***Auto connect when project is opened*** in the dialog ***KNX project settings***. Thus, you do not have to establish the connection manually.

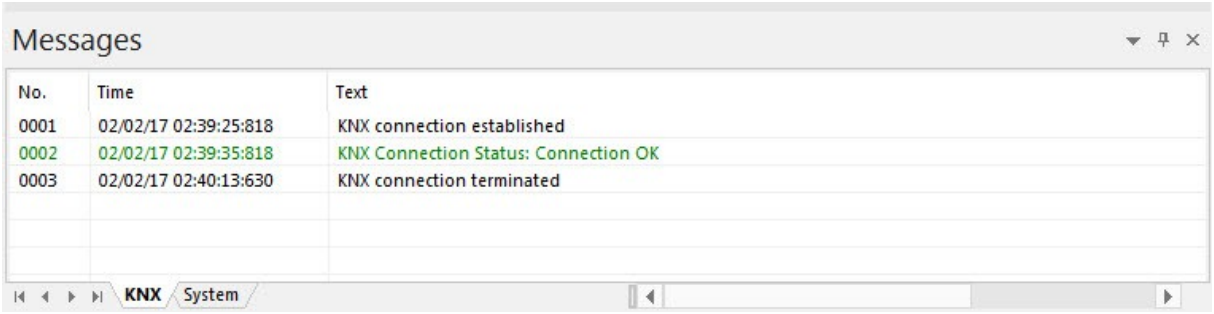
Reconnect after connection failure:

If this option is enabled, the system reconnects automatically to the bus in case of a connection failure.

4.14 Connect to KNX Bus

To start the connection to the KNX bus, choose the menu item ***Options*** and click ***Start KNX***. Alternatively, click on the symbol in the toolbar.

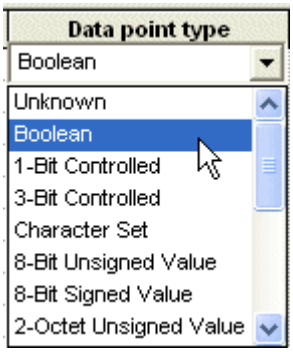
To interrupt the connection to the bus, choose the menu item ***Options*** and click ***Stop KNX*** or click the symbol in the toolbar.

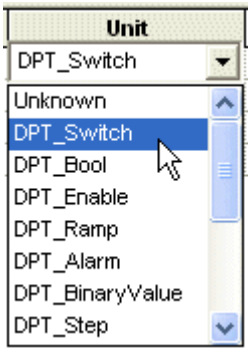


You can see the results of these operations and possibly already received telegrams in the message list.

4.15 Send Telegrams Manually

In the KNX group addresses grid make sure that the columns ***Data point type***, ***Unit***, and ***Rights*** contain the proper value for each KNX group address.





If the KNX group address is properly defined and the write access is allowed, a value can be entered in the column ***Value***. Alternatively, right-click in the value column and select the desired value in the context menu (picture below).

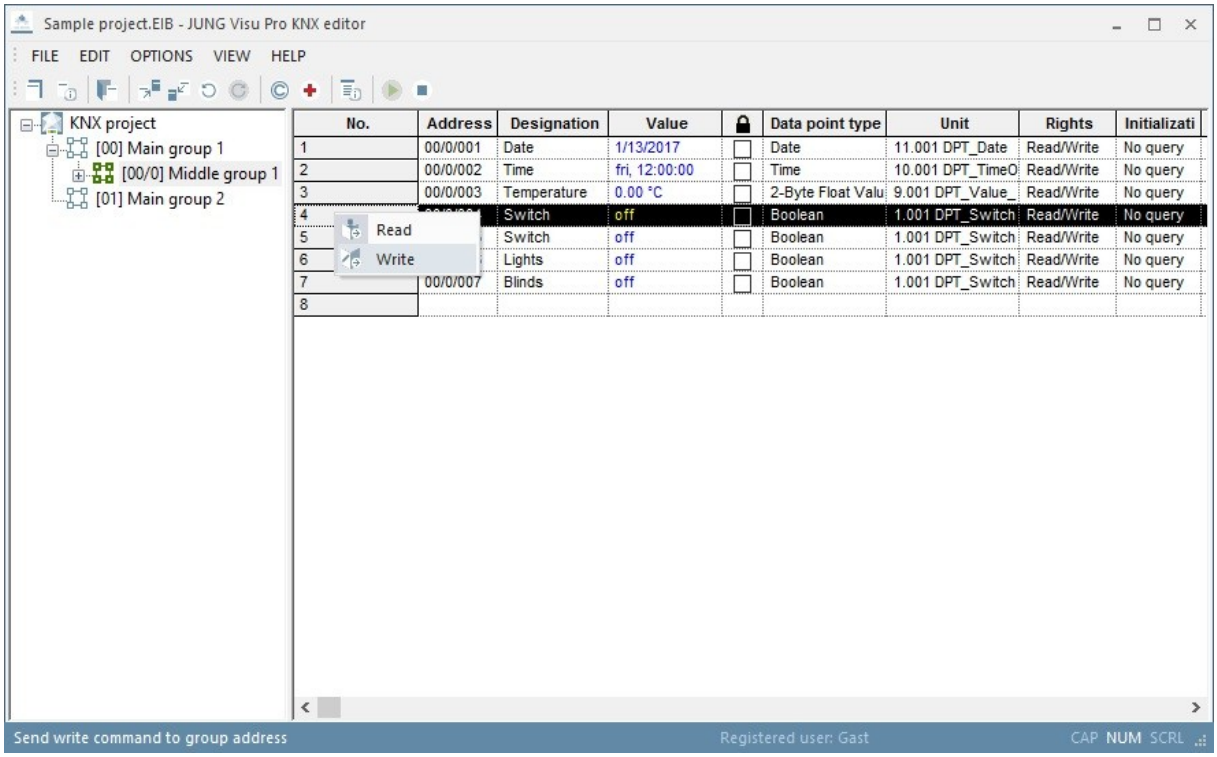
No.	Address	Designation	Value		Data point type	Unit	Rights
1	00/0/001	Date	1/13/2017	☐	Date	11.001 DPT_Date	Read/Write
2	00/0/002	Time	fri, 12:00:00	☐	Time	10.001 DPT_TimeO	Read/Write
3	00/0/003	Temperature	0.00 °C	☐	2-Byte Float Valu	9.001 DPT_Value_	Read/Write
4	00/0/004	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write
5	00/0/005	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write
6	00/0/006	Lights	off	☐	Boolean	1.001 DPT_Switch	Read/Write
7	00/0/007	Blinds	off	☐	Boolean	1.001 DPT_Switch	Read/Write
8							

0 off
1 on

For binary values (type Boolean) use ***0*** for ***off*** and ***1*** for ***on***. For other types just enter the value. The KNX editor will perform the required data conversions.

You can enter an analog value, for example **25.65** for the data point type 2 Octet Float. When the value is to be sent, the content of this column is read, converted into a KNX telegram on the basis of the specified data point type and sent via the bus.

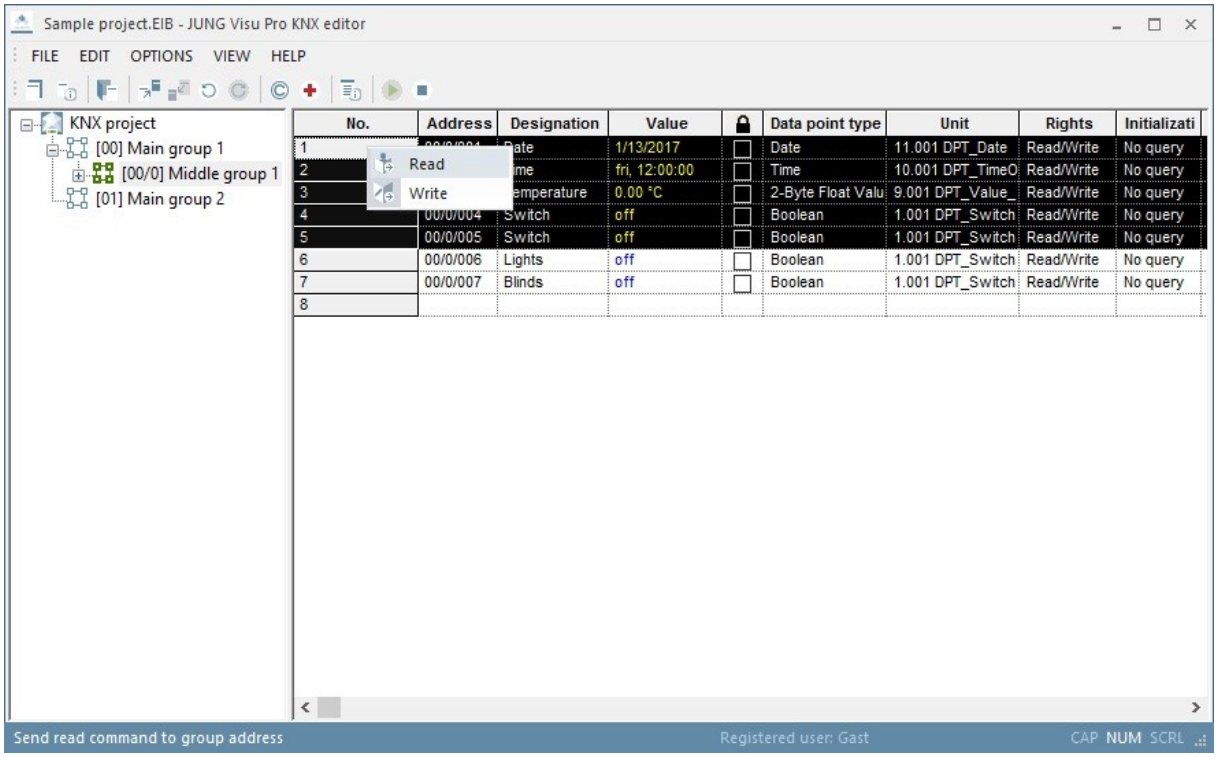
To send the value via the bus, select the entire row of the group address with a left click on the ***No.*** column in the row of the group address. Then right-click and select the desired command, ***Read*** for sending or ***Write*** for querying the value.



No.	Address	Designation	Value		Data point type	Unit	Rights	Initializati
1	00/0/001	Date	1/13/2017	☐	Date	11.001 DPT_Date	Read/Write	No query
2	00/0/002	Time	fri, 12:00:00	☐	Time	10.001 DPT_TimeO	Read/Write	No query
3	00/0/003	Temperature	0.00 °C	☐	2-Byte Float Valu	9.001 DPT_Value_	Read/Write	No query
4		Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
5		Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
6		Lights	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
7	00/0/007	Blinds	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
8								

After the command has been performed, the telegrams are shown in the message list.

To send several commands at the same time, click the ***No.*** column. Then perform a right-click and select the desired command in the context menu.



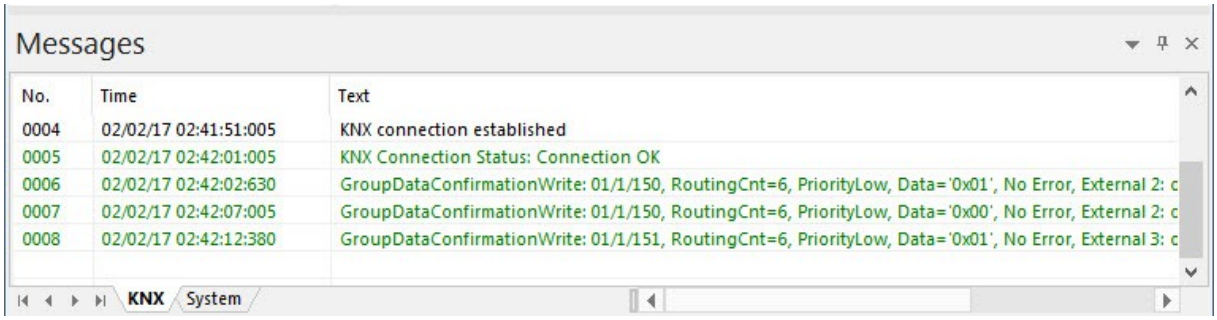
No.	Address	Designation	Value		Data point type	Unit	Rights	Initializati
1		Date	1/13/2017	☐	Date	11.001 DPT_Date	Read/Write	No query
2		ime	fri, 12:00:00	☐	Time	10.001 DPT_TimeO	Read/Write	No query
3		emperature	0.00 °C	☐	2-Byte Float Valu	9.001 DPT_Value_	Read/Write	No query
4		Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
5	00/0/005	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
6	00/0/006	Lights	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
7	00/0/007	Blinds	off	☐	Boolean	1.001 DPT_Switch	Read/Write	No query
8								

4.16 Message List

The message list shows all KNX telegrams sent and received. When sufficient information regarding KNX data point type and unit has been supplied, the telegrams will be displayed in plain text.

To toggle the visibility of the message list, choose the menu item ***View*** and click ***Messages***. Alternatively, click the symbol in the toolbar.

Choose the menu item ***View*** and click ***Show last message*** to specify that the message list scrolls automatically to the last message.



Meaning of the colors in the message list:

Green: Sent and acknowledged telegrams.

Blue: Received telegrams.

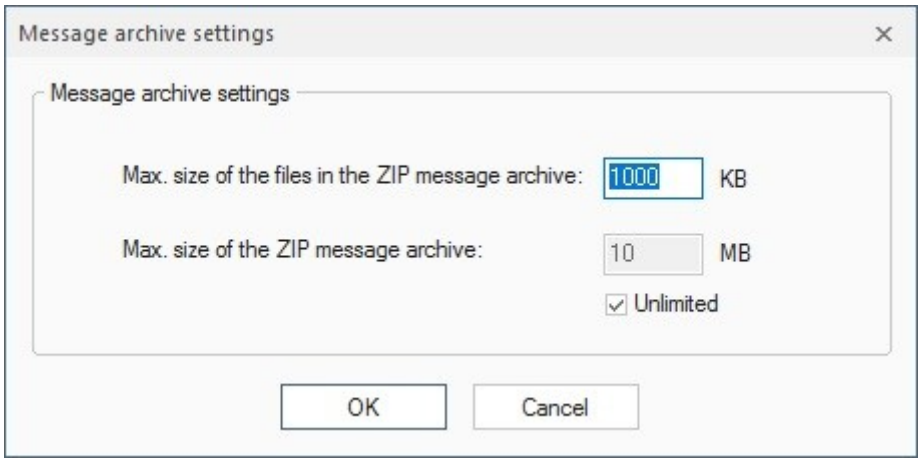
Red: Status or error messages.

Grey: Read requests.

4.17 Logging KNX Messages into Archives

Messages displayed in the message list can be automatically saved in a text file. Choose the menu item ***Options*** and click ***Save KNX telegrams*** to activate or deactivate the automatic logging.

To configure the settings for the message archive, choose the menu item ***Options*** and click ***KNX archive settings***. The dialog ***Message archive settings*** opens.



When the telegram log file reaches a size of about 1 Megabyte, it will be closed and saved into the zip archive **EibMessages.zip** at the directory of the KNX project**:**

(...\stations\local\projects\<projectname>.EIB\EibMessages.zip).

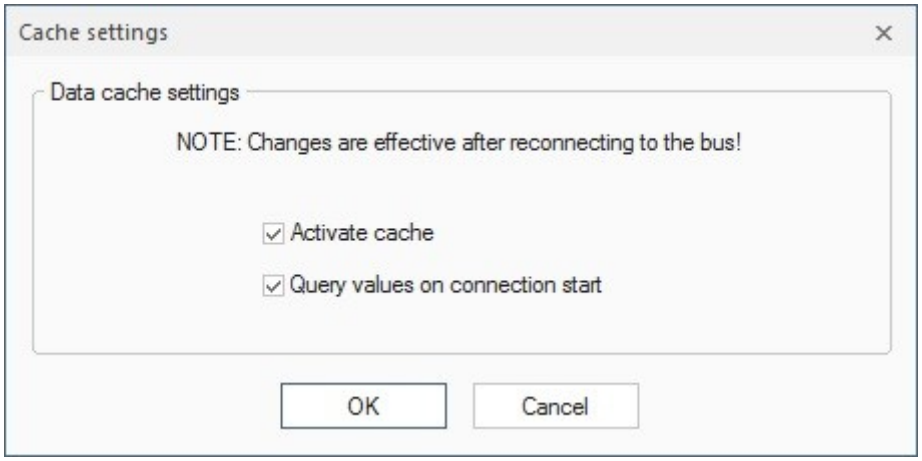
A new log file will be created. The log file will also be closed and saved in the zip archive when the KNX bus has been disconnected.

4.18 Data Cache Settings

The KNX editor comprises a data cache to optimize access to current values of KNX group addresses. Optionally, the cache can be initialized once by querying the bus, but if the values of KNX group addresses are available in the cache, it will update the cache from listening to the KNX without additional queries.

> ***Note:*** *The cache will always be cleared, when the bus is disconnected.*

Choose the menu item ***Options*** and click ***Cache settings***. The dialog Cache settings opens.



Check boxes within the dialog:

Activate cache: Select this check box to activate the cache.

Query values on connection start: When this check box is selected, the cache will query values for KNX group addresses whenever a connection to the KNX bus is established.

> ***Attention:*** *Querying values may produce unexpected results when the access rights for the KNX group addresses have not been properly configured.*
>
> ***Therefore check the settings of the access rights before enabling this option.***

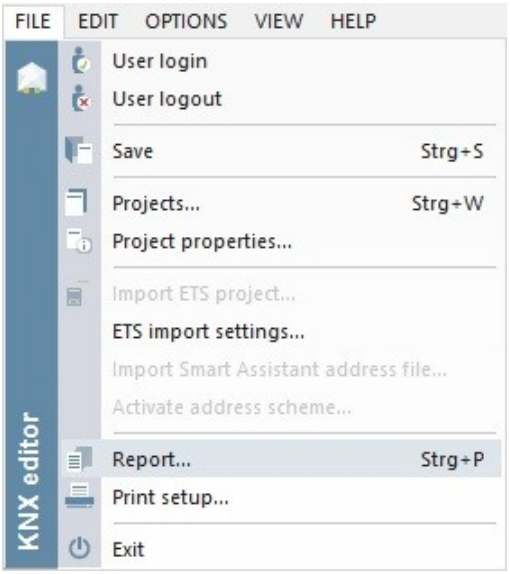
Choose the menu item ***Options*** and click ***Clear cache*** to clear the cache manually.

5 Reports

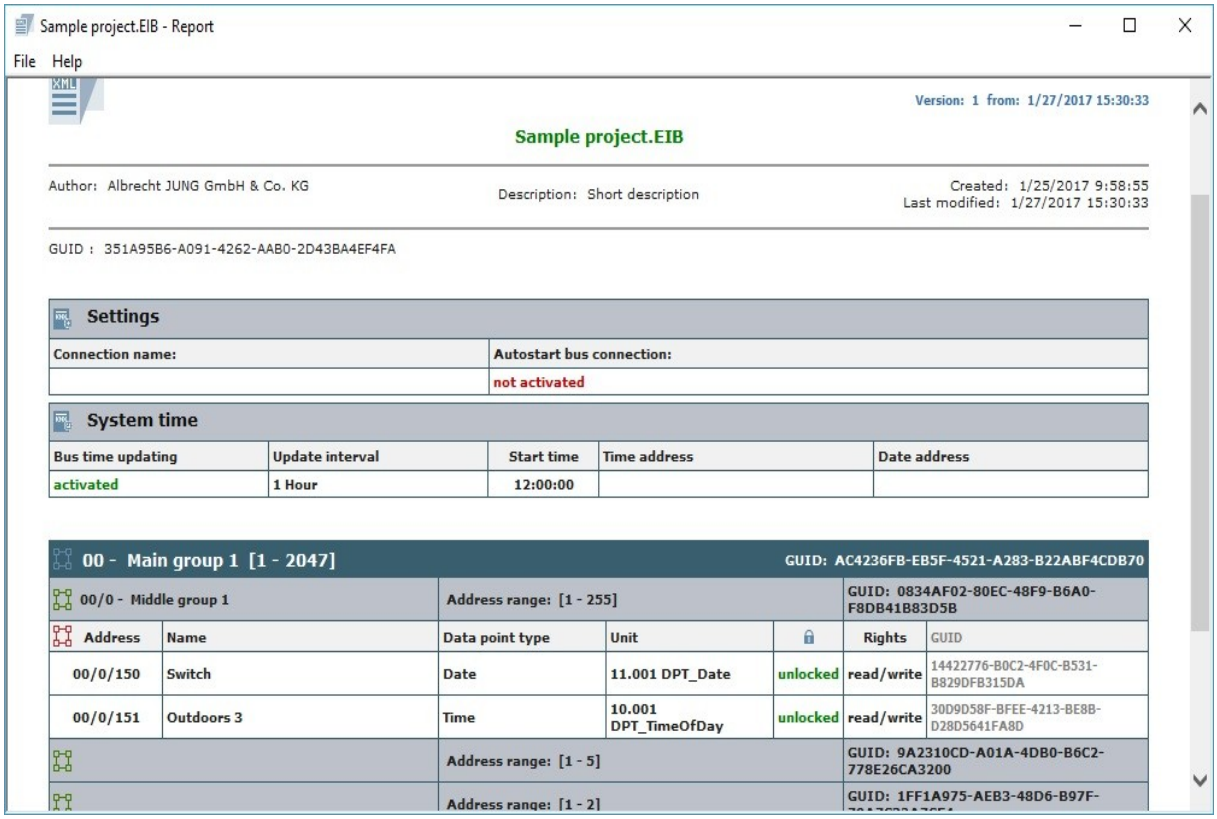
- A report can be created which gives information about the process variables contained in the project and their properties.
- Reports can be printed out (with print preview).

5.1 Create Report

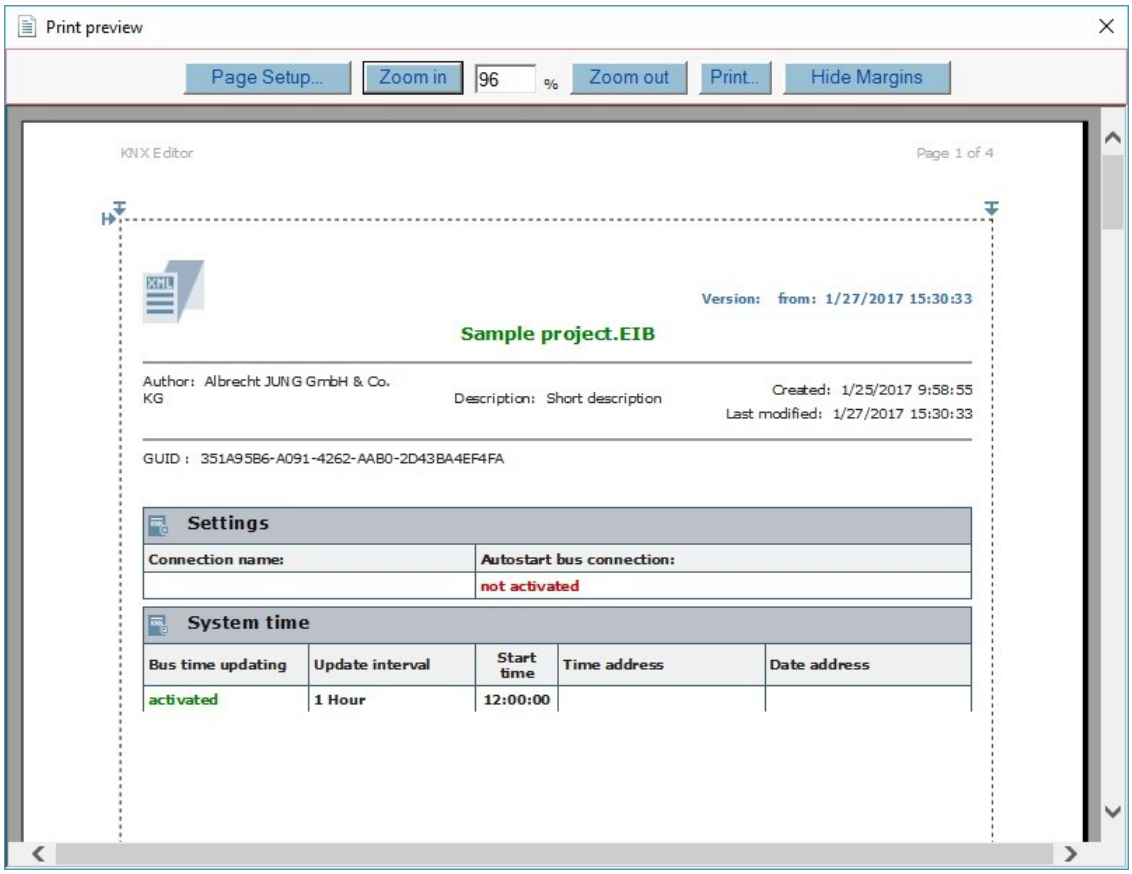
To create a report for the current project, choose the menu item ***File*** and click ***Report***. The dialog ***Create report*** opens. Select the desired options for the report generation and click the button ***Create***.



The data points in the project may be sorted by either names or group addresses.


Sample project.EIB - Report
File Help
Version: 1 from: 1/27/2017 15:30:33
Sample project.EIB
Author: Albrecht JUNG GmbH & Co. KG
Description: Short description
Created: 1/25/2017 9:58:55
Last modified: 1/27/2017 15:30:33
GUID : 351A95B6-A091-4262-AAB0-2D43BA4EF4FA
Settings
Connection name:
Autostart bus connection:
not activated
System time
Bus time updating
Update interval
Start time
Time address
Date address
activated
1 Hour
12:00:00
00 - Main group 1 [1 - 2047]
GUID: AC4236FB-EB5F-4521-A283-B22ABF4CDB70
00/0 - Middle group 1
Address range: [1 - 255]
GUID: 0834AF02-80EC-48F9-B6A0-F8DB41B83D5B
Address
Name
Data point type
Unit
Rights
GUID
00/0/150
Switch
Date
11.001 DPT_Date
unlocked
read/write
14422776-B0C2-4F0C-B531-B829DFB315DA
00/0/151
Outdoors 3
Time
10.001 DPT_TimeOfDay
unlocked
read/write
30D9D58F-BFEE-4213-BE8B-D28D5641FA8D
Address range: [1 - 5]
GUID: 9A2310CD-A01A-4DB0-B6C2-778E26CA3200
Address range: [1 - 2]
GUID: 1FF1A975-AEB3-48D6-B97F-

5.2 Print Preview



The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these ↦,↧ markers the user has visual control over the margin settings.

JUNG Visu Pro
KNX Editor 2.0

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Process Interface....3
1.4 Import from ETS....3
1.5 Internet Explorer, DirectX and .Net Framework....4

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....11

4 FUNCTIONS....11

4.1 KNX Project Organizer....11
4.1.1 Create KNX Project....12
4.1.2 Open KNX Project....13
4.1.3 Delete KNX Project....13
4.1.4 Rename KNX Project....14
4.2 Edit Project Properties....15
4.3 Convert KNX Editor 1.0 Project....16
4.4 Installations....17
4.4.1 Create Installation....17
4.4.2 Create Multiple Installations....18
4.4.3 Duplicate Installation....19
4.4.4 Duplicate Multiple Installations....19
4.4.5 Configure Installation....21
4.4.6 Configure Installations and Interfaces (Expert)....22
4.4.7 Configuration Report....24
4.5 Group Addresses....25
4.5.1 Create Main Group....26
4.5.2 Create Middle Group....26
4.5.3 Create Group Address Range....27
4.5.4 Create Group Addresses....29
4.5.5 Edit Group Addresses....30
4.6 Start Groups....32
4.7 Request Groups....35
4.8 Report Groups....39
4.8.1 Report Formatting....44
4.9 KNX System Time....45
4.10 Synchronize PC Time with KNX Time....46
4.11 Application of KNX Data Types....52
4.12 ETS Import of KNX Group Addresses....55

4.12.1 Create ETS Group Addresses Print Report....56
4.12.2 Import OPC Export File from the ETS....56
4.12.3 Import of ETS Export Files into the KNX Editor....58
4.12.4 ETS Import Settings....62
4.13 Import of Smart Assistant Address Files....64
4.14 Activate Address Scheme....64
4.15 KNX Gateway Settings....66
4.16 Connect to KNX Bus....67
4.17 Send Telegrams Manually....67
4.18 Message List....69
4.19 Logging KNX Messages into Archives....70
4.20 Data Cache Settings....71
5 REPORTS....72
5.1 Create Report....72
5.2 Print Preview....73

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz
Main memory	1024 MB
Free disk space (additional disk space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Process Interface

The connection to the KNX bus requires the installation of a suitable version of the FALCON driver of the KNX Association and the appropriate interfaces.

Optionally, the visualization system is equipped with an OPC (OLE for Process Control) client, so that instead of or additionally to KNX, OPC servers - which are available for a multitude of automation systems – can be used for the connection to the process.

1.4 Import from ETS

ETS5	YES, use of ETS, project export
ETS4	YES, use of ETS, project export
ETS3, ETS2 version 1.3	YES, use of ETS, OPC export
ETS2 versions 1.1, 1.2 and 1.3	YES, use of print reports redirected to file
Earlier ETS versions	NO

1.5 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The .NET Framework has to be activated via Windows features in Windows 8 and later. The JVP KNX Gateway, which is used to connect the KNX editor to KNX also needs the .NET Framework 4.6.2.

2 Introduction

After the installation of the **visualization system** a program group is created in the Start Menu. The default settings can be adopted or changed manually during installation.

Click the program symbol to start the control panel of the **visualization system**. In order to clearly distinguish between the different applications, different colors were used. The icon of the **KNX editor 2.0** contains a blue colored bar .

To start the individual function modules, double-click the symbol of the respective modules.

The **KNX editor 2.0** is part of the **visualization system**. It is used to create KNX projects which serve as a basis for the communication with the bus and for the operation of visualization projects.

First steps:

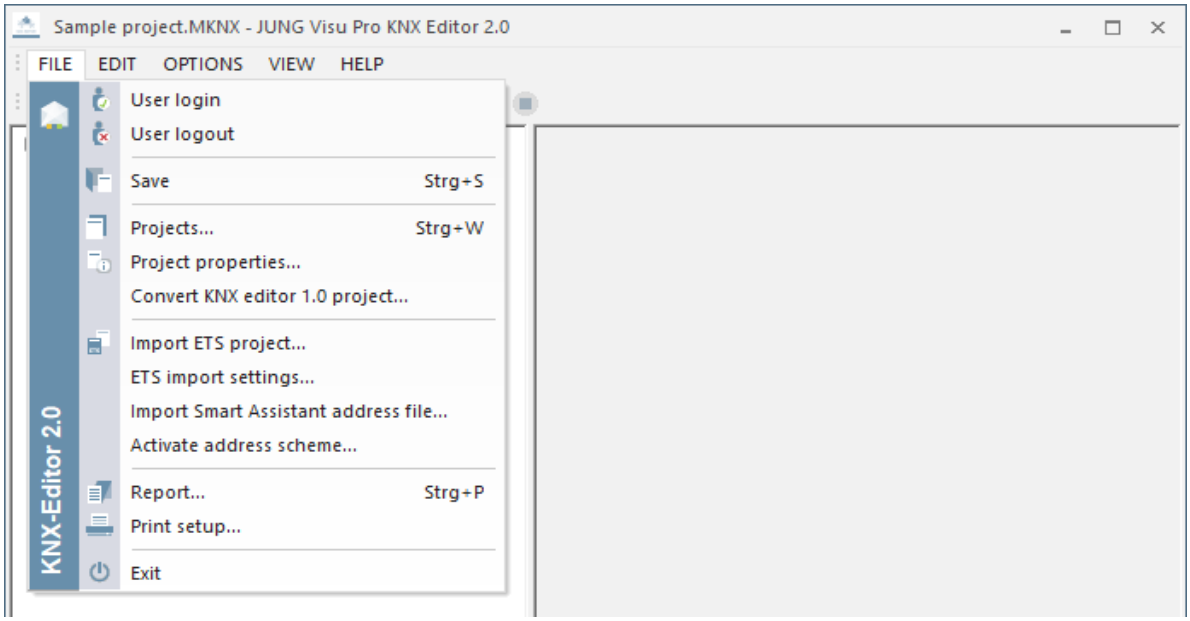
1. Create a KNX project.
2. Create an installation.
3. Create group addresses manually or import them from an ETS project.
4. Specify information concerning the data points: data point type, unit and rights.
5. Save the settings.
6. Connect the project to the KNX bus.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

3 User Interface

3.1 Menu

Menu item *File*



User login: Opens a dialog to log in a user to the system.

User logout: Logs current user out of the system.

Save: Saves the project.

Projects: Opens a dialog to create and maintain KNX projects.

Project properties: Opens a dialog where you can edit general project properties.

Convert KNX editor 1.0 project...: Converts KNX projects 1.0 to 2.0.

Import ETS project: Opens a dialog to import an ETS project.

ETS import settings: Opens a dialog to set standard datapoint types that are used when the typ is not specified in the ETS import file.

Import Smart Assistant address file: Opens a dialog to import Smart Assistant address files.

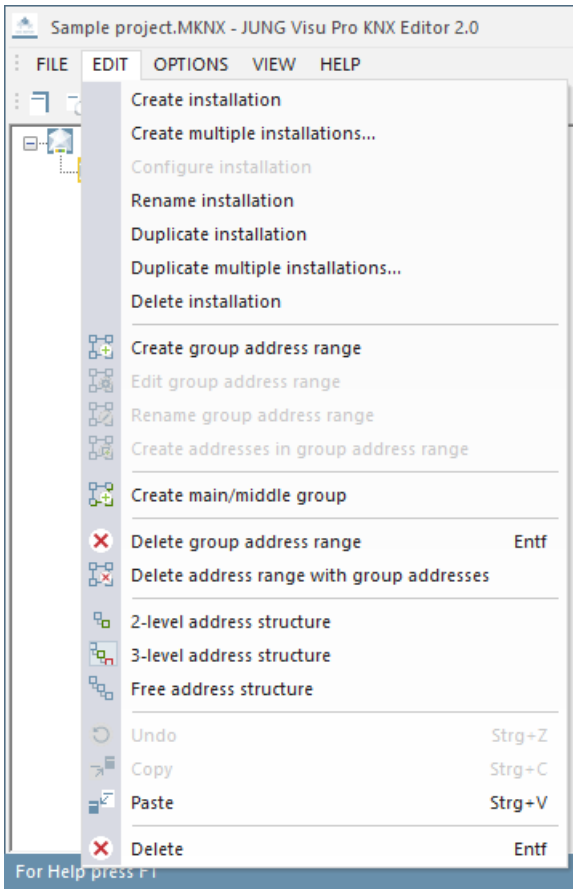
Activate address scheme: Opens a dialog to import an ETS project.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Quits the KNX editor. Shortcut ALT+F4.

Menu item *Edit*



Create installation: Creates a new installation in the KNX project.

Create multiple installations...: Creates multiple installations in the KNX project at once.

Configure installation: Opens the dialog for editing the selected installation.

Rename installation: Opens the input field to rename the selected installation.

Duplicate installation: Creates a copy of the selected installation.

Duplicate multiple installations: Creates multiple copies of the selected installation.

Delete installation: Deletes the selected installation and all underlying group addresses.

Create group address range: Opens the dialog ***New address range***.

Edit group address range: Opens the dialog for editing the range limits of the selected group.

Rename group address range: Opens the input field of the selected group address range.

Create addresses in group address range: Opens the dialog ***Create group addresses***.

Create main/middle group: Opens the dialog ***New main group*** or ***New middle group***.

Delete group address range: Deletes only the selected address range. The group addresses contained in this address range will be placed in the address range above. If there is none, the group addresses will be deleted as well.

Delete address range with group addresses: Deletes the selected address range with all address ranges and group addresses included in that address range.

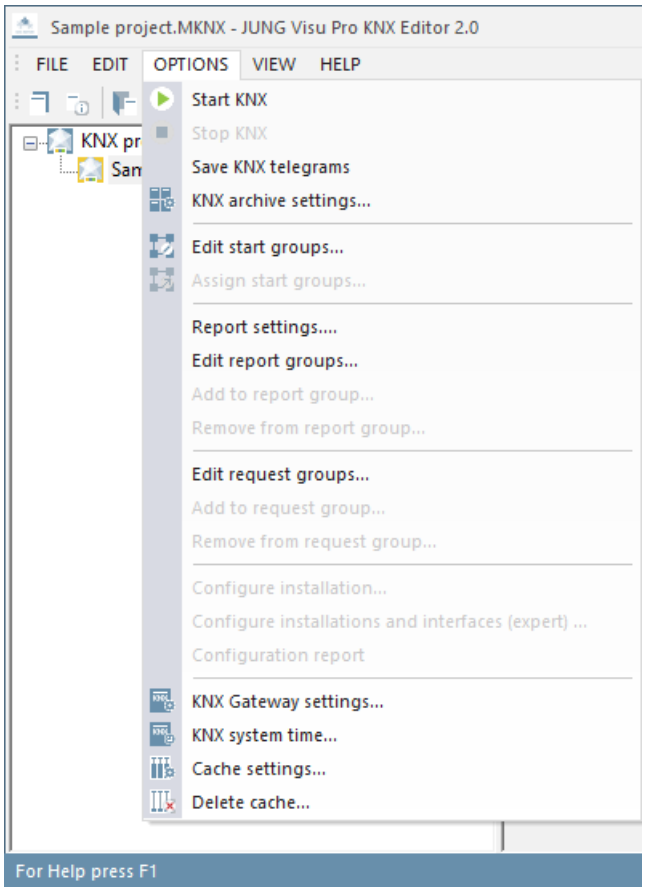
2-level address structure: Switches to the 2-level address structure for group addresses (00/0001).

3-level address structure: Switches to the 3-level address structure for group addresses (00/0/001).

Free address structure: Switches to the free address structure for group addresses (00001).

Standard editing options: Undo, Copy, Paste, Delete.

Menu item *Options*



Start/stop KNX: Connects to or disconnects from the KNX bus.

Save KNX telegrams: Saves KNX telegrams to a file.

KNX archive settings...: Opens a dialog to edit archive settings.

Edit start groups...: Opens a dialog to edit start groups.

Assign start groups...: Opens a dialog to assign start groups.

Report settings: Opens the dialog to edit the report groups.

Edit report groups: Opens the dialog to edit report groups.

Add to report group: Opens the dialog for the selection of report groups.

Remove from report group: Opens the dialog for the selection of report groups.

Edit request groups: Opens the dialog to edit request groups.

Add to request group: Opens the dialog for the selection of request groups.

Remove from request group: Opens the dialog for the selection of request groups.

Configure installation...: Opens the ***Installation settings*** dialog.

Configure installations and interfaces (expert)...: Opens the ***KNX interfaces overview*** dialog.

Configuration report: Creates an information overview for the interfaces that are assigned to the installations.

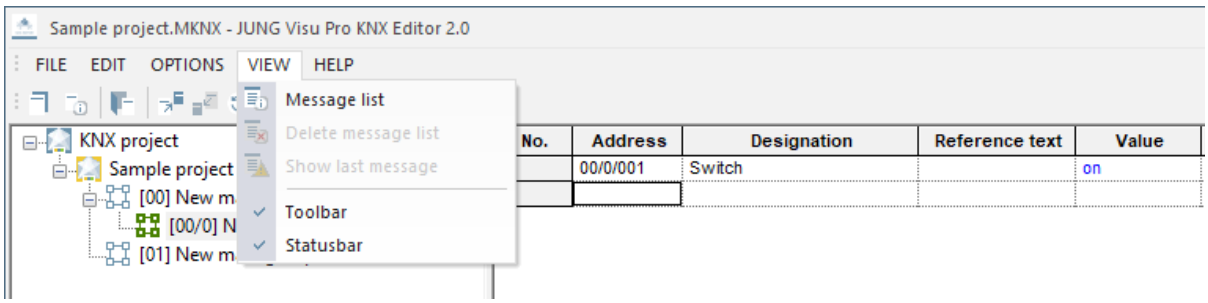
KNX Gateway settings: Opens a dialog to configure the connection to KNX.

KNX system time: Opens a dialog to update the KNX clock from KNX.

Cache settings: Opens a dialog to configure the data cache.

Delete cache: Deletes the cache.

Menu item *View*

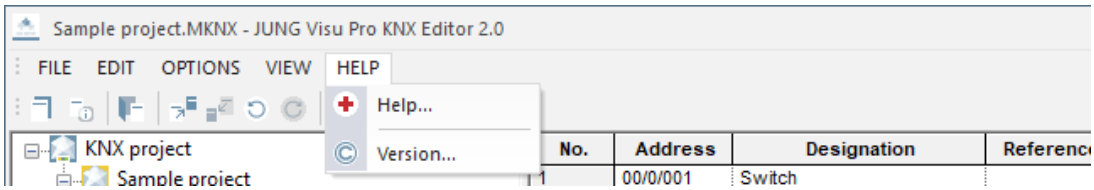


Message list: Toggles the visibility of the messages list at the bottom of the editor.

Delete message list: Deletes the messages from the message window. This operation does not affect the saving of telegrams to an archive file.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

Menu item *Help*



Help...: Opens help file.

Version...: Shows version and copyright information.

3.2 Toolbar

Projects management: Opens a dialog to create and maintain KNX projects.
Project properties: Opens a dialog to edit general project properties.
Save: Saves the project.
Copy: Copies selected items to clipboard.
Paste: Pastes items from clipboard.
Undo: Undo last action.
Redo: Redo last action, i.e. reverse an undo operation.
Version: Shows version and copyright information.
Help: Opens help file.
Messages: Toggles visibility of message list at the bottom of the editor.
Start KNX: Connects to KNX bus.
Stop KNX: Disconnects from KNX bus.

4 Functions

4.1 KNX Project Organizer

In the dialog ***KNX Project Organizer*** you can create new projects and open, close, delete or rename existing projects.

In the project organizer you can use the buttons on the sidebar or the commands under the menu item ***Project***.

The project organizer provides the following options:

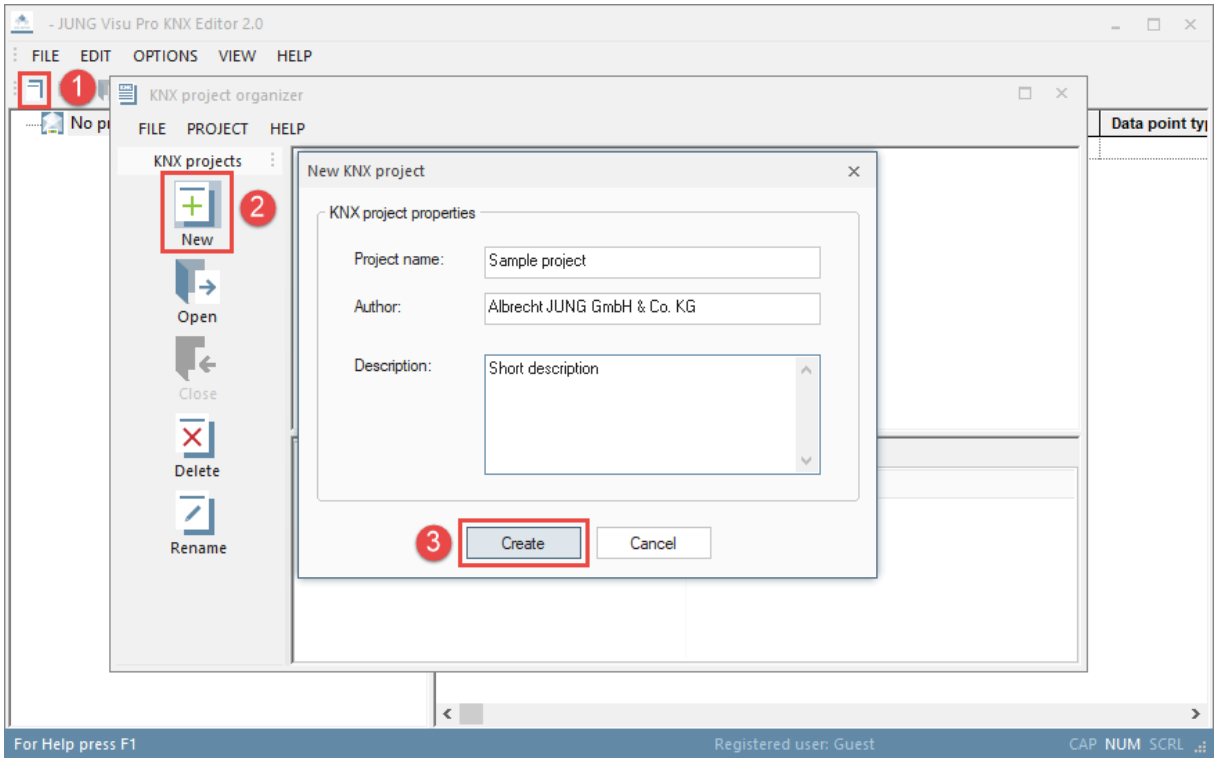
- **New**: Create a new KNX project.
- **Open**: Open an existing project in the KNX editor.
- **Close**: Close the currently opened KNX project.
- **Delete**: Delete the selected project.
- **Rename**: Rename a project.

4.1.1 Create KNX Project

The dialog ***KNX project organizer*** opens automatically when the KNX editor is started.

To start it manually in the KNX editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

To create a new KNX project, click the button ***New*** on the sidebar or choose the menu item ***Project*** and click ***New***. The dialog ***New KNX project*** opens.



Enter the data of the new project:

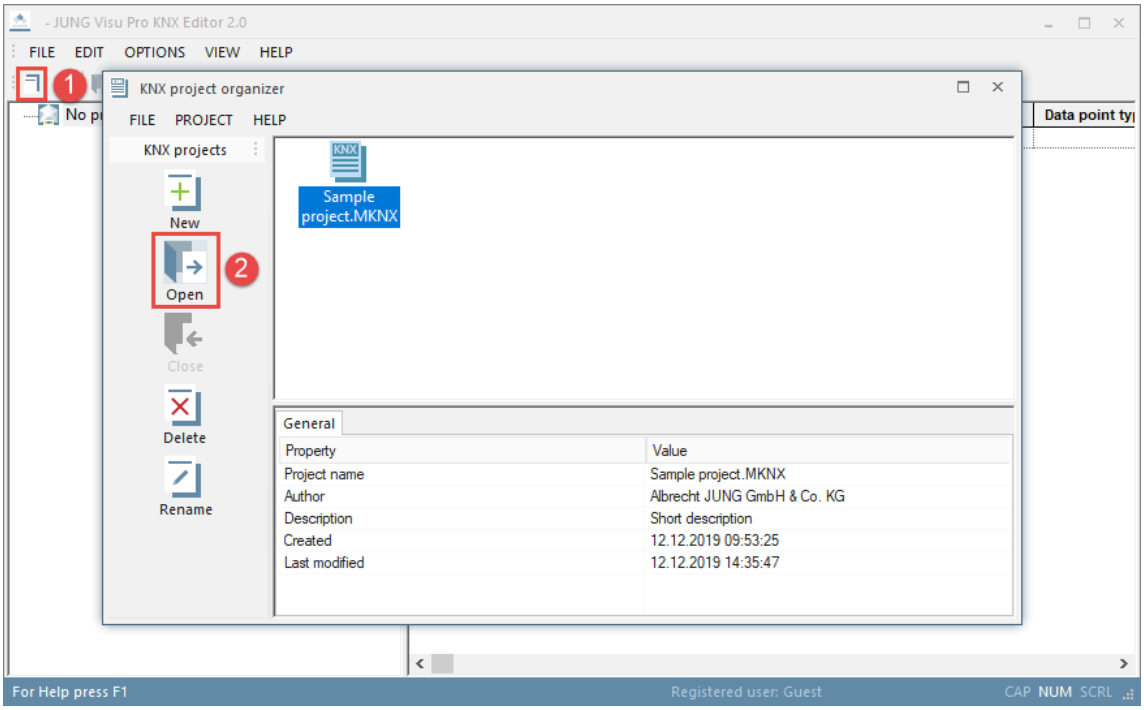
Project name: Enter a unique name for the KNX project.

Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or edited later.

Create: Press the button ***Create*** to commit the entries and to finish the creation of the project.

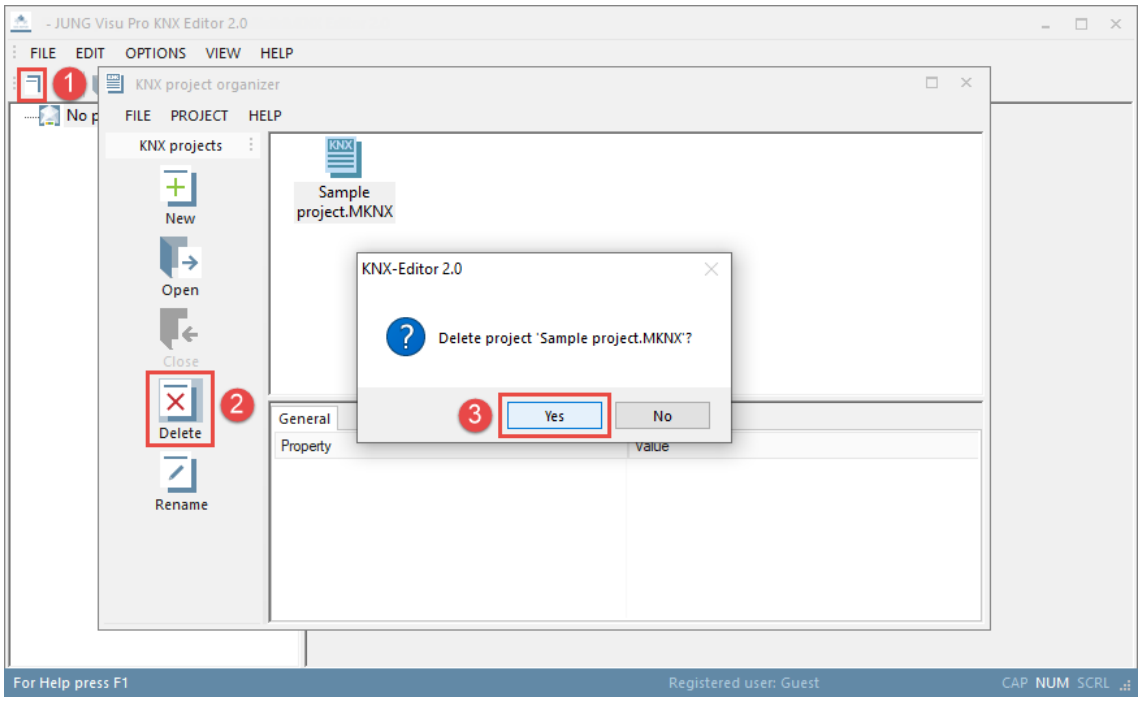
4.1.2 Open KNX Project

To load a KNX project into the KNX editor, double-click the project symbol in the project list or select the desired project and click the button ***Open*** in the sidebar.



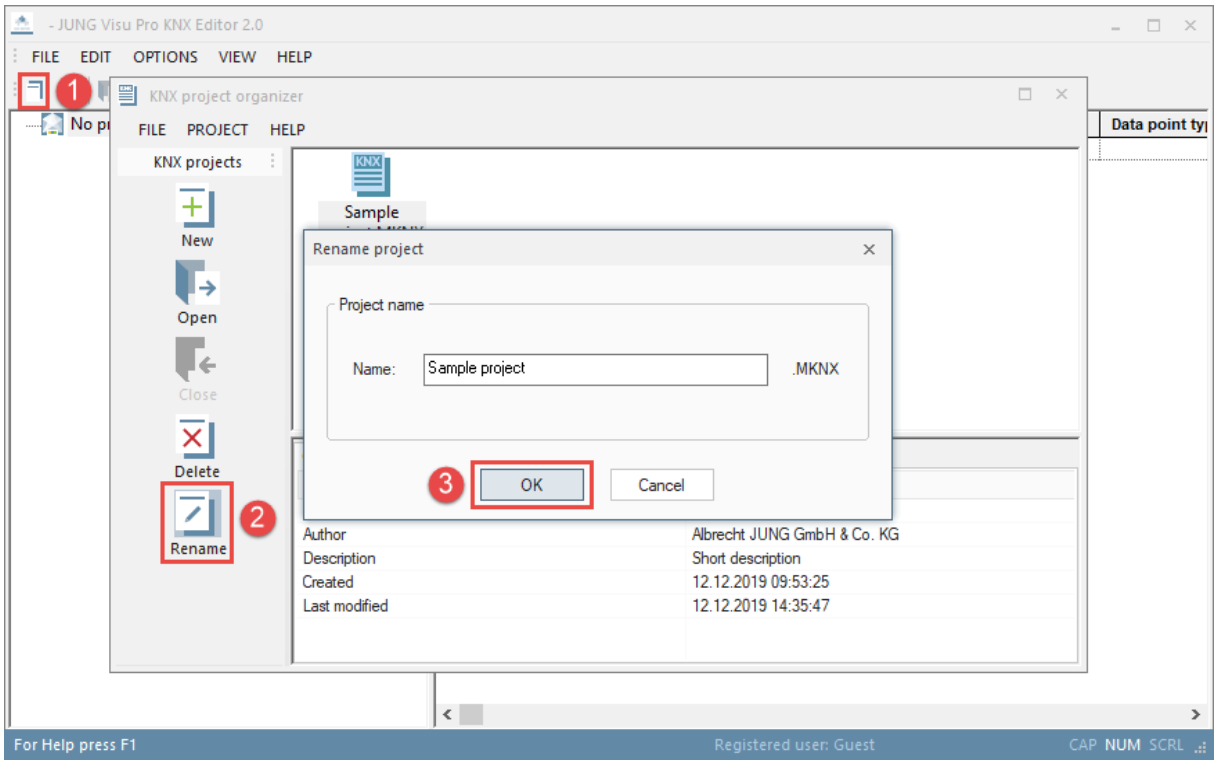
4.1.3 Delete KNX Project

To delete a project, select the project and click the button ***Delete*** in the sidebar.



4.1.4 Rename KNX Project

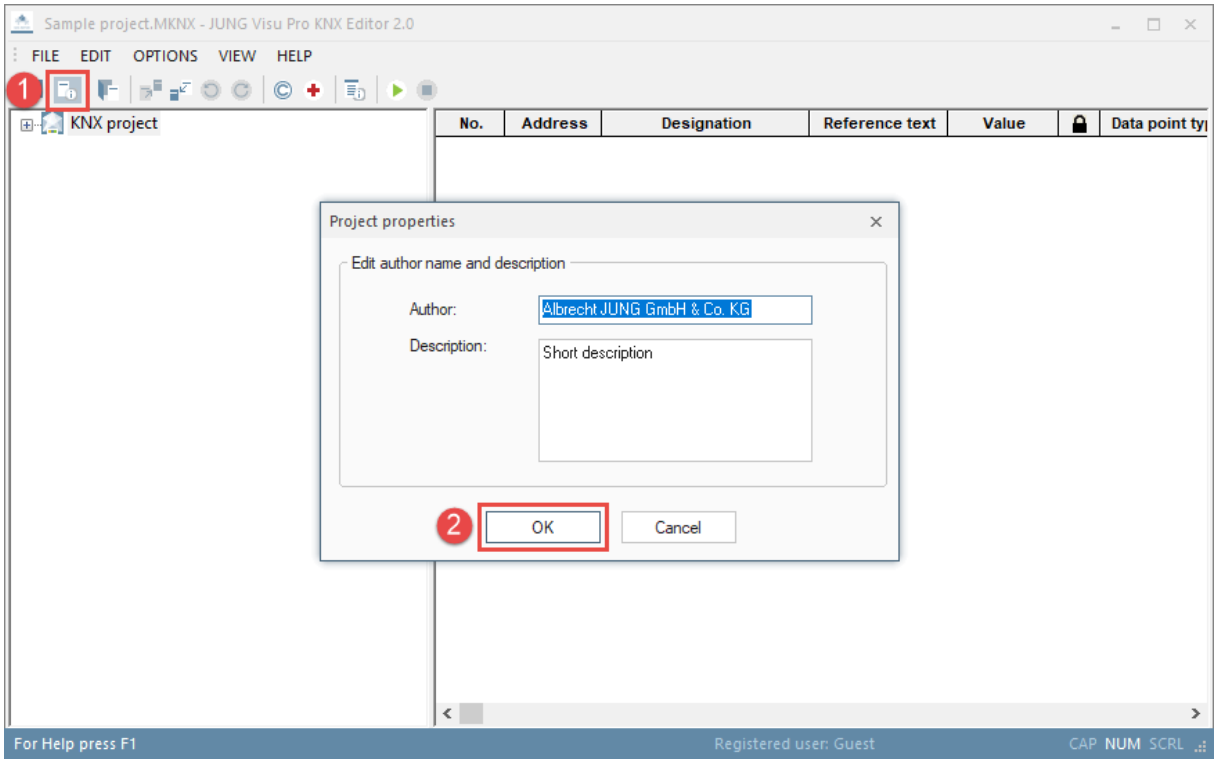
To rename a project, select the desired project and click the button ***Rename*** in the sidebar. The dialog ***Rename project*** opens.



Enter in the name field the new project name and click the button ***OK***. The dialog is closed and the new project name is adopted. The extension ***.EIB*** is automatically added to the project name.

4.2 Edit Project Properties

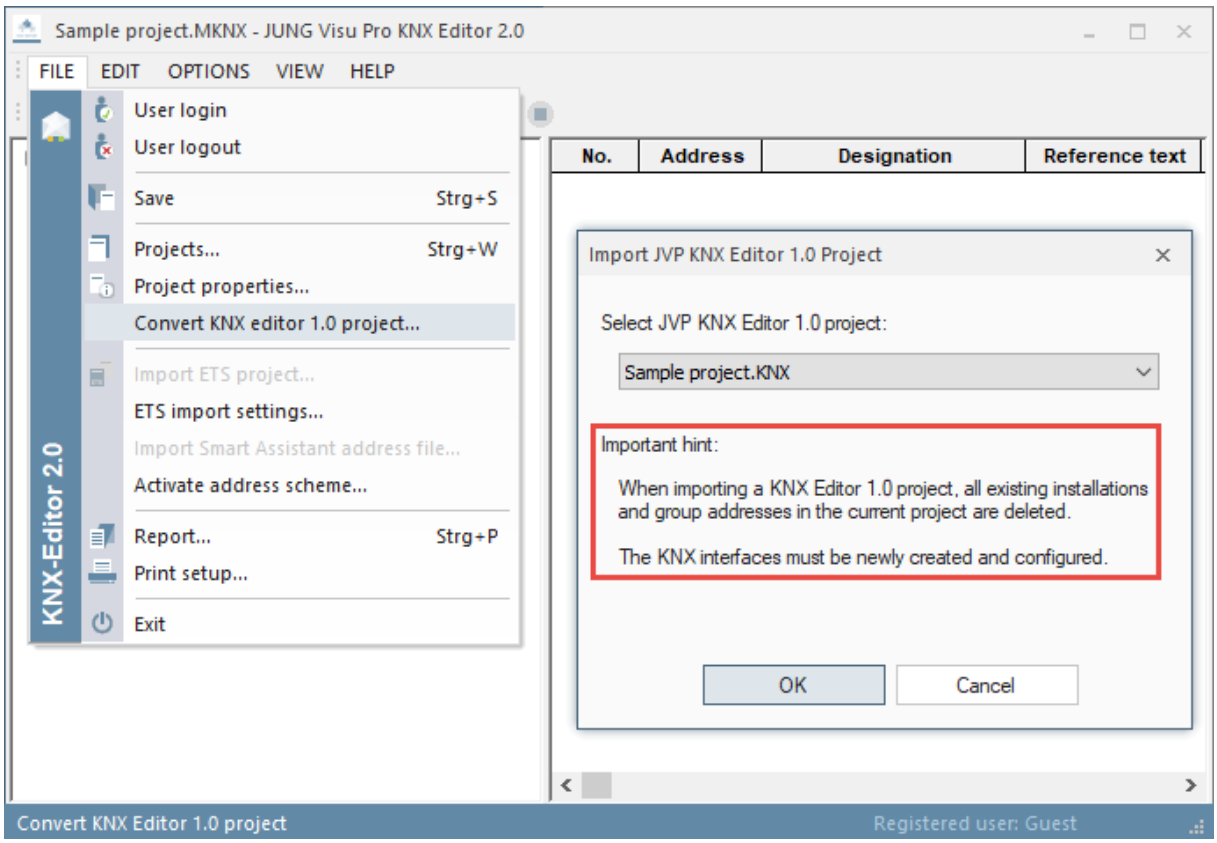
Choose in the KNX editor the menu item ***File*** and click ***Project properties*** or click the symbol in the toolbar. The dialog ***Project properties*** opens where you can edit general properties of the project.



Edit the specifications author and description and click the button ***OK*** to commit the changes and to close the dialog.

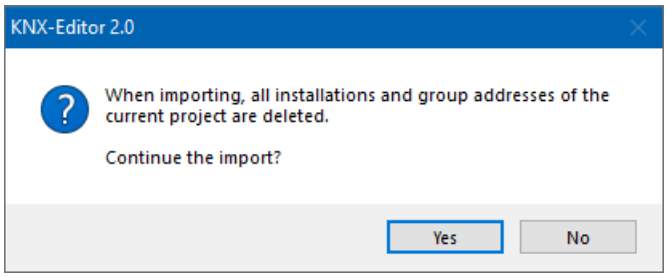
4.3 Convert KNX Editor 1.0 Project

The KNX editor 2.0 enables the user to connect several KNX projects in one editor. Older KNX projects 1.0 can be converted to 2.0. First create a new project in the KNX editor 2.0. Then click on ***Convert KNX editor 1.0 project...*** in the menu ***File***.

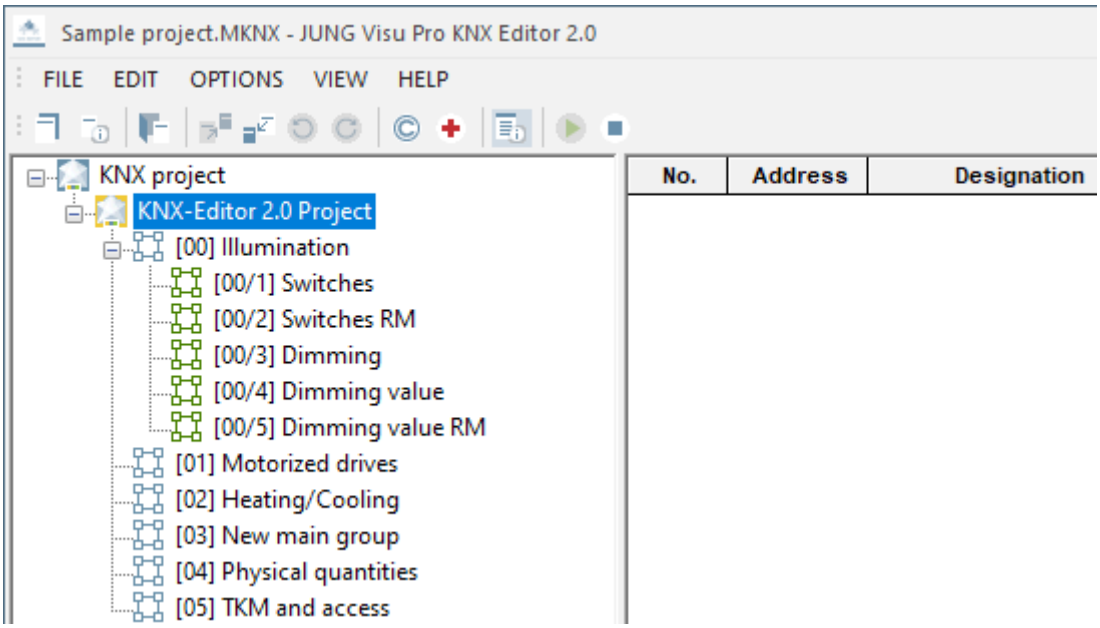


Select the KNX project 1.0, which should be converted and click ***OK.***

Attention:	Please note the advice. A confirmation prompt appears. If a project that already contains installations and group addresses is open during this process, these are deleted during the import of the KNX project 1.0.



Confirm the security prompt by clicking ***Yes***. Now all group addresses of the KNX project 1.0 are available in your newly created KNX project 2.0 in an installation.



To import the new project into the process model without having to reconnect the data points of your visualization, it is also necessary to convert the project in the process model. Detailed instructions can be found in the manual of the process model.

4.4 Installations

An installation contains a complete KNX project with possible 32 main groups. Every installation has its own KNX interface to communicate separately with the bus.

4.4.1 Create Installation

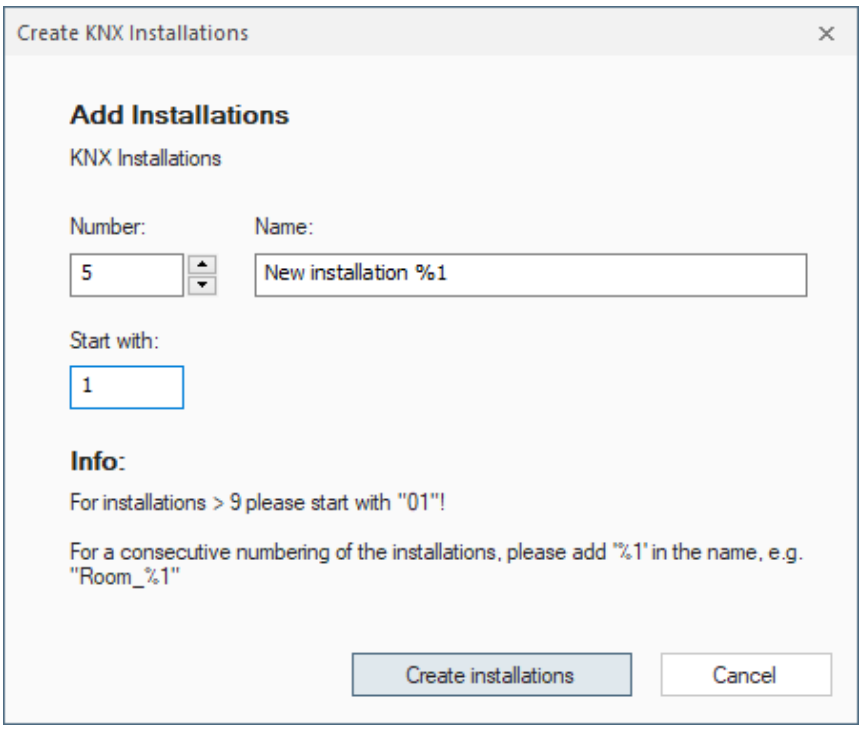
There are two ways to create an installation:

- By clicking on ***Create installation*** in the menu ***Edit***.
- Right click on the project symbol in the directory tree to open the context menu and choose ***Create installation***.

4.4.2 Create Multiple Installations

To create multiple installations at once,

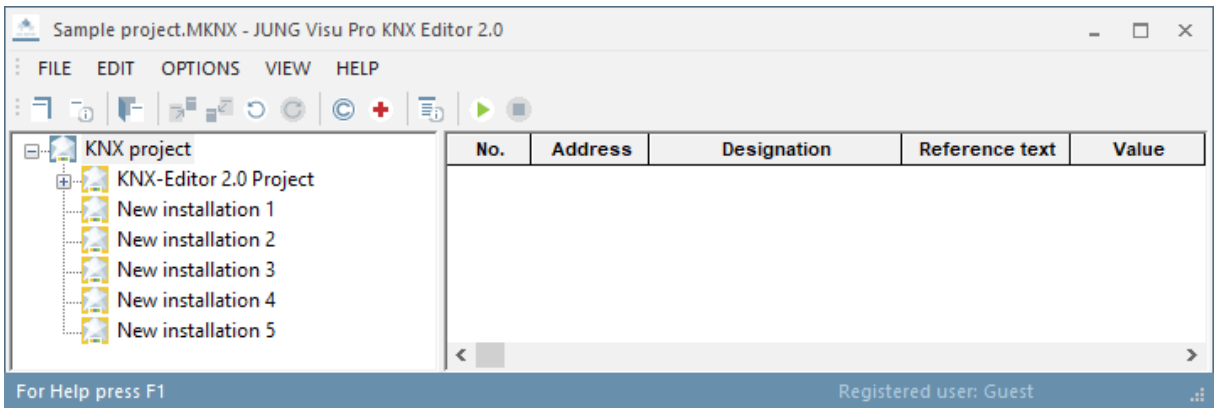
- Click on ***Create multiple installations...*** in the menu ***Edit*** or
- Right click on the **project symbol** in the directory tree to open the context menu and choose ***Create multiple installations...*** The following dialog opens:



Number: Enter the number of installations you want to create. A maximum of 50 installations can be created in one operation.

Name: Enter a name. By adding the placeholder %1, the newly created installations are serially numbered counting from the start value.

Start with: The initial digit of the consecutive numbering.



4.4.3 Duplicate Installation

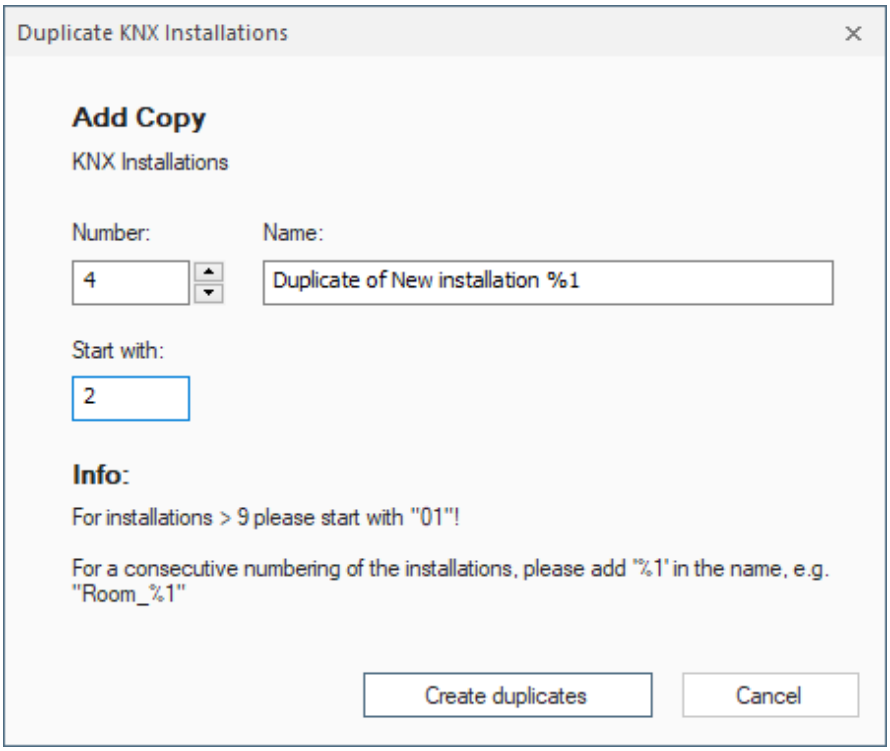
To duplicate an installation,

- Select the installation you want to duplicate and click ***Duplicate installation*** in the menu ***Edit*** or
- Right click on the installation in the directory tree you want to duplicate to open the context menu and choose ***Duplicate installation***.

4.4.4 Duplicate Multiple Installations

To create multiple copies of the selected installation,

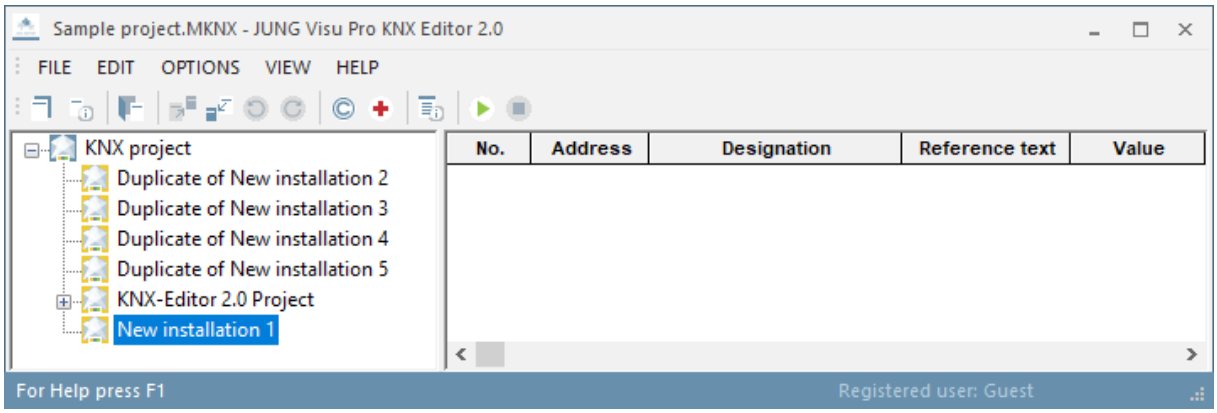
- Select the installation you want to duplicate and click ***Duplicate multiple installations...*** in the menu ***Edit*** or
- Right click on the installation in the directory tree you want to duplicate to open the context menu and choose ***Duplicate multiple installations...***



Number: Enter the number of copies you want to create. A maximum of 50 copies can be created in one operation.

Name: Enter a name. By adding the placeholder %1, the newly created installations are serially numbered counting from the start value.

Start with: The initial digit of the consecutive numbering.


Sample project.MKNX - JUNG Visu Pro KNX Editor 2.0
FILE
EDIT
OPTIONS
VIEW
HELP
KNX project
Duplicate of New installation 2
Duplicate of New installation 3
Duplicate of New installation 4
Duplicate of New installation 5
KNX-Editor 2.0 Project
New installation 1
No.
Address
Designation
Reference text
Value
For Help press F1
Registered user: Guest

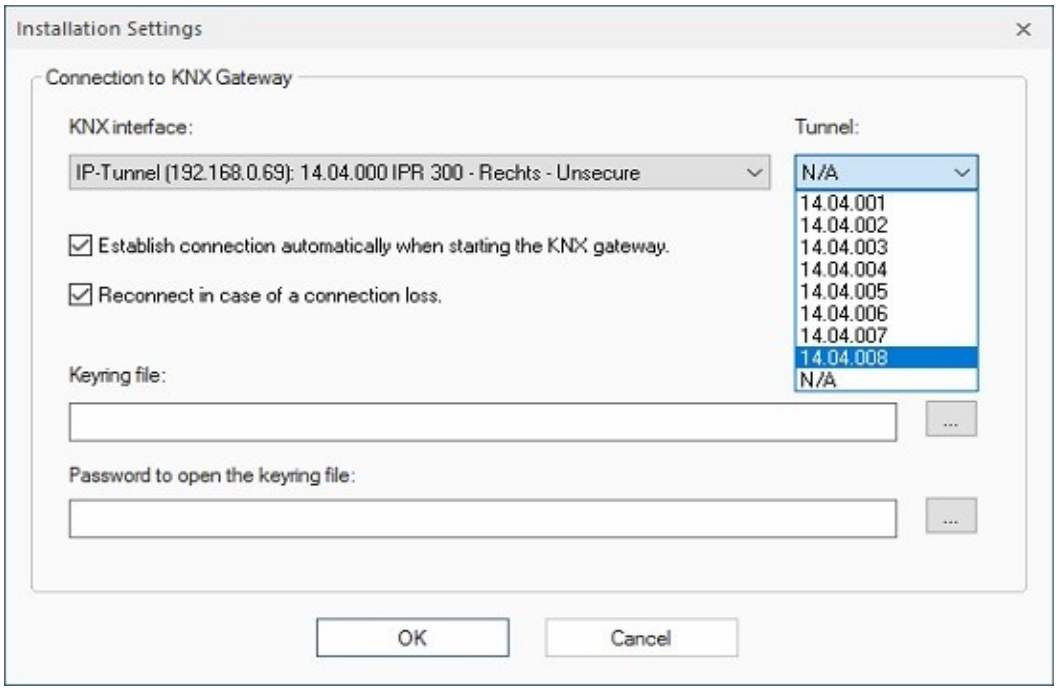
4.4.5 Configure Installation

To edit an installation, the KNX editor has to be connected to the JUNG KNX Gateway at least once. For this purpose KNX has to be started. For more information see chapter 4.14 **KNX Gateway settings**.

There are two ways to edit an installation:

- Select an installation in the directory tree and then click on ***Configure installation*** in the menu ***Edit***.
- Right click on an installation in the directory tree to open the context menu and choose ***Configure installation***.

The Dialog ***Installation Settings*** will open.



In this dialog you can select a KNX interface for the selected installation. It will show all available interfaces, which are provided by the JUNG KNX Gateway. Select one of the interfaces and click on OK.

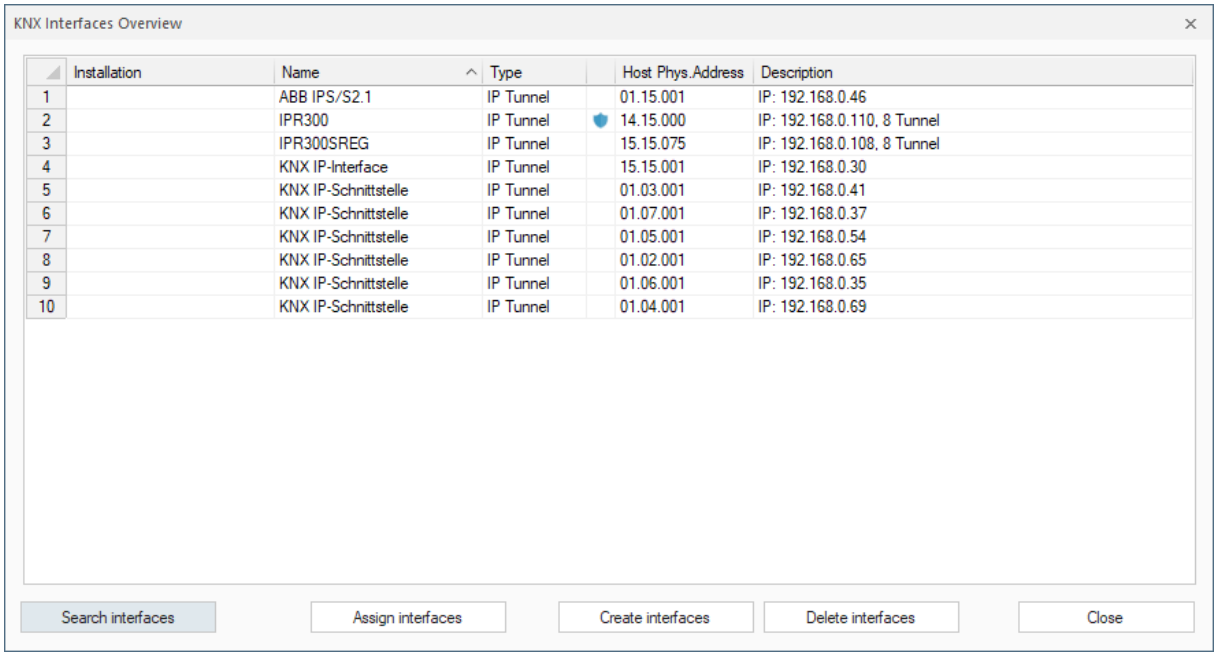
Reconnect in case of a connection loss:	If the connection to the selected KNX interface cannot be established, a new attempt will be tried every 60 seconds.
Keyring file:	This option is only available if the selected router is running in secured mode. The file is generated in the ETS and is used to establish a secure connection.
Password to open the keyring file:	Enter the password, which is also defined in the ETS.

4.4.6 Configure Installations and Interfaces (Expert)

To configure installations and interfaces, the KNX editor must first be connected to a KNX Gateway. For this purpose KNX has to be started. For more information, refer to chapter 4.14 **KNX Gateway Settings**.

Click ***Configure installations and interfaces (expert)...*** in the menu ***Options***.

The following dialog opens:

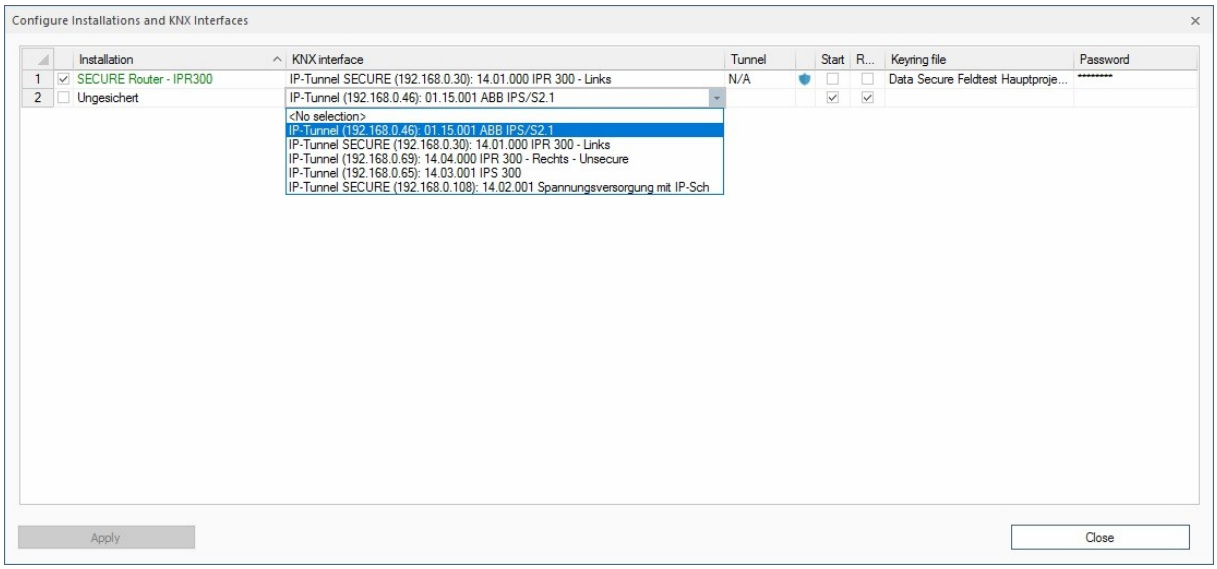


The overview shows all interfaces found by the KNX Gateway.

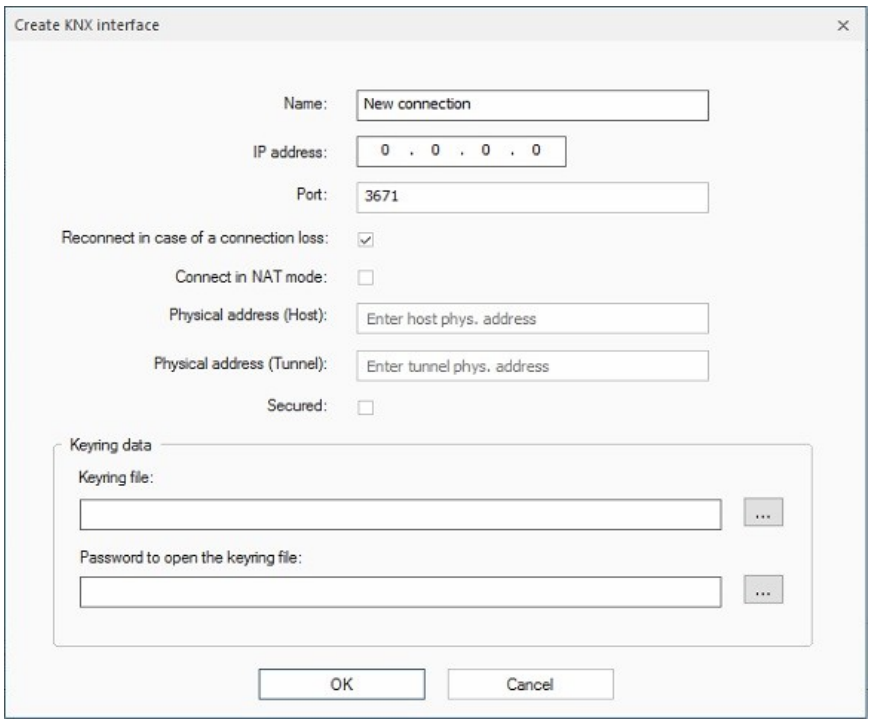
Installation:	Displays the installation assigned to the interface.
Name:	Name of the interface.
Type:	Type of interface.
	Displays an encrypted connection.
Host Phys. Address:	Displays the physical address of the host.
Description:	Provides more detailed information about the interface.

Search interfaces: Tells the KNX gateway to search again for interfaces in the network.

Assign interfaces: Click on the button to open a dialog in which all installations of the current project are listed. An interface can be assigned to the respective installation in the column **KNX interface**. If this is an IP tunnel, a tunnel can also be assigned in the column **Tunnel**. If the router supports IP Secure, a **keyring file** with the corresponding **password** must be entered. The keyring file of the interface and the password can be found in the ETS. If the router should be connected automatically when the KNX gateway starts, check the box at **Start**. Confirm the settings with ***Apply***.



Create interfaces: It is possible to manually set up an interface that is not found by the discovery. Click on the ***Create interface*** button to open the following dialog:



Name:	Name of the interface.
IP address:	IP address of the interface.
Port:	Port of the interface.
Reconnect in case of a connection loss:	If the connection to the selected KNX interface cannot be established, a new attempt will be tried every 60 seconds.

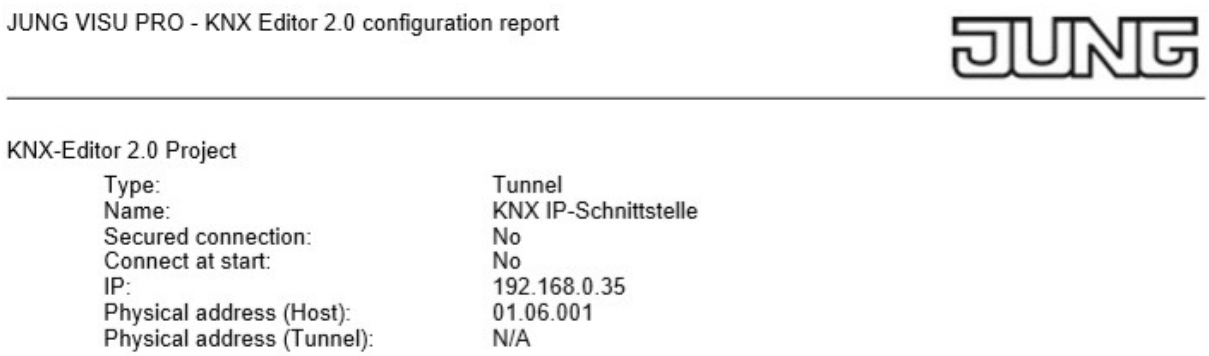
Connect in NAT mode:	Should not be used to connect to unsecured interfaces over the internet.
Physical address (Host):	Physical address of the interface.
Physical address (Tunnel):	Physical address of the tunnel, which shall be used.
Secured:	Check, if the interface is in secured mode.
Keyring file:	This option is only available if the selected router is running in secured mode. The file is generated in the ETS and is used to establish a secure connection.
Password to open the keyring file:	Enter the password, which is also defined in the ETS.

Delete interfaces: Click on the line of the interface you want to delete (it is now highlighted) and press the ***Delete interface*** button.

Close: Closes the window.

4.4.7 Configuration Report

Click ***Configuration report*** in the ***Options*** menu, to obtain an information overview of the interfaces that are assigned to an installation in the project.



JUNG VISU PRO - KNX Editor 2.0 configuration report

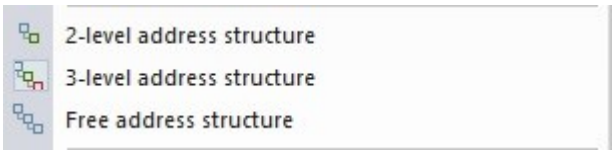
KNX-Editor 2.0 Project

Type:	Tunnel
Name:	KNX IP-Schnittstelle
Secured connection:	No
Connect at start:	No
IP:	192.168.0.35
Physical address (Host):	01.06.001
Physical address (Tunnel):	N/A

4.5 Group Addresses

You can create group addresses manually or import them from ETS projects via the import function. More details on the ETS import will be provided in one of the subsequent chapters. In the following, the manual creation of group addresses is described. Manually created group addresses can not be exported to the ETS.

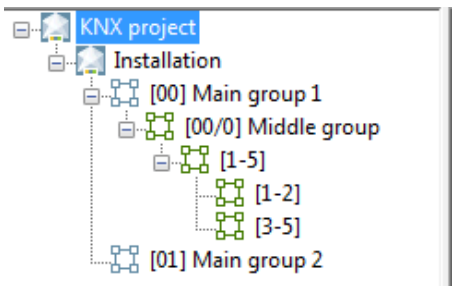
Under the menu item ***Edit*** you can choose between 3 different structures of group addresses:



2-level address structure: The group addresses are arranged in a 2-level scheme; main group address and group address *(00/0001)*.

3-level address structure: The group addresses are arranged in a 3-level scheme; main group address, middle group address and group address *(00/0/001)*.

Free address structure: Free display of the group addresses *(00001)*.

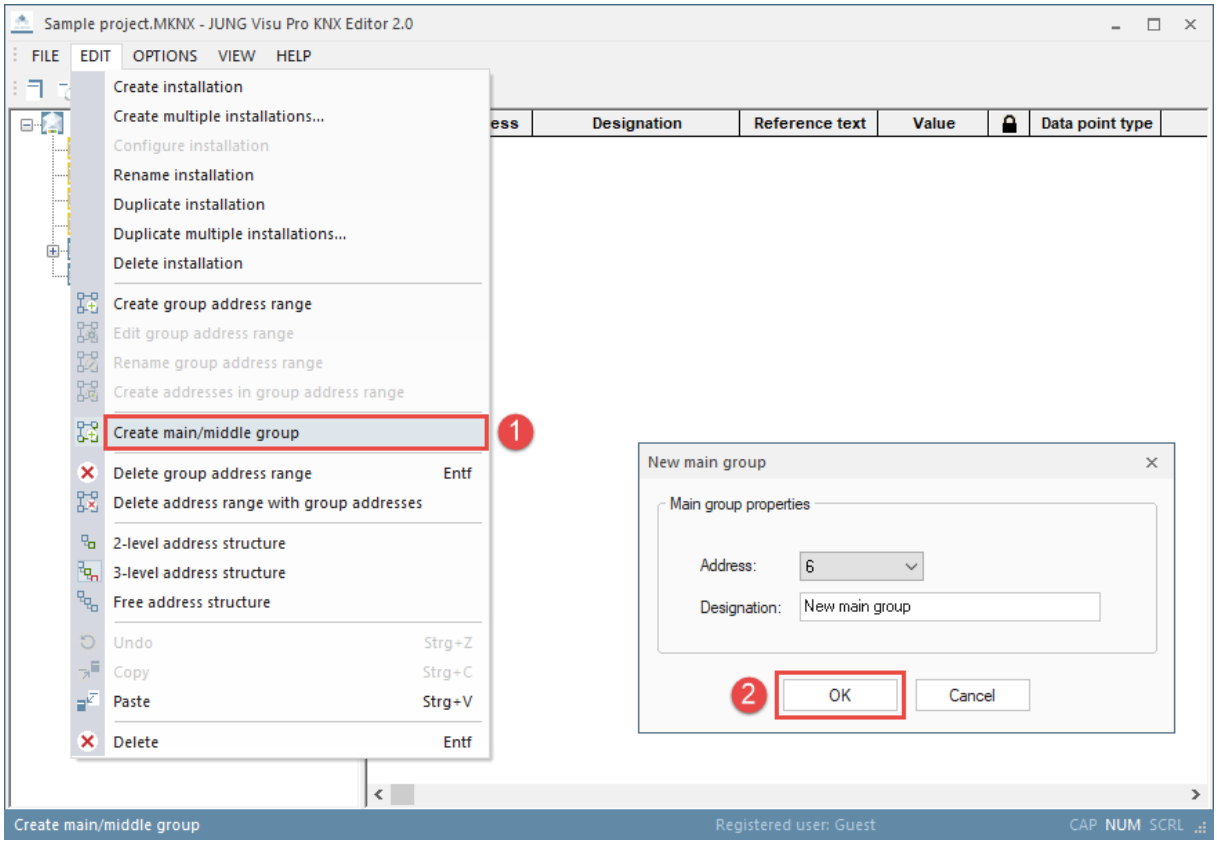


In the left window of the editor you can see the KNX projects with their respective groups (if already created) arranged in a structure tree.

4.5.1 Create Main Group

To create a main group,

- select an installation in the directory tree, choose the menu item ***Edit*** and click ***Create main/middle group*** or
- right-click an installation in the directory tree and click in the context menu the option ***Create main/middle group***.



The dialog ***New main group*** opens. Enter the main group address and a name for the main group. Click the button ***OK*** to commit the data and to close the dialog.

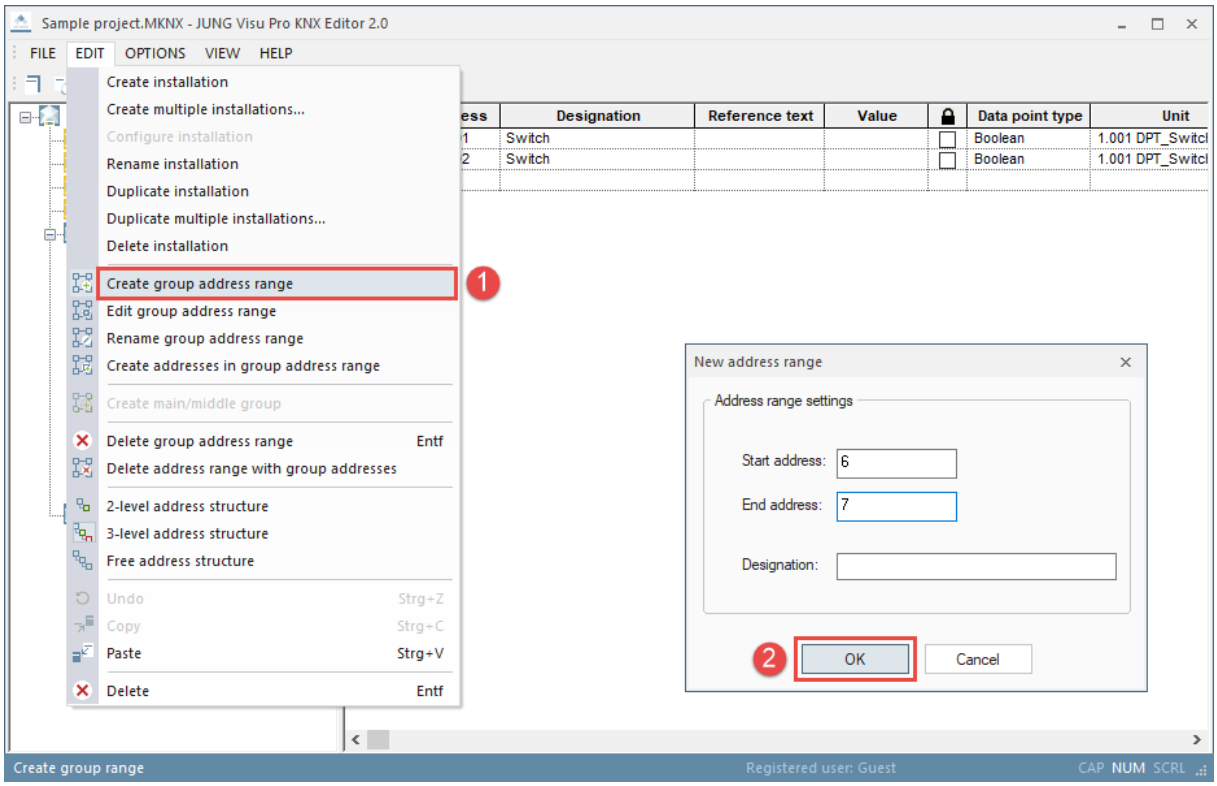
Use this method to create further KNX main group addresses.

4.5.2 Create Middle Group

To create a middle group, choose the menu item ***Edit*** and click ***Create main/middle group***. Alternatively, right-click the main group symbol in the directory tree and click in the context menu the option ***Create main/middle group***.

4.5.3 Create Group Address Range

To create a group address range, select in the directory tree an installation or group for which you want to create the address range. Choose the menu item ***Edit*** and click ***Create group address range***. Alternatively, right-click an installation or group in the directory tree and click in the context menu the option ***Create group address range***.



The dialog ***New address range*** opens. Enter the start and the end address (range limits) and, where appropriate, a name for the selected group address range. Click the button ***OK*** to commit the entry.

The specified address range will then be assigned to the corresponding or the superordinate address range.

Change group address range:

Select in the directory tree the group address range you wish to edit. Choose the menu item ***Edit*** and click ***Edit group address range***. A dialog opens where you can change the range limits of the selected group address range. Enter the new range limits and click the button ***OK*** to commit the change.

Rename group address range:

Select in the directory tree the group address range you wish to rename. Choose the menu item ***Edit*** and click ***Rename group address range***. The input field of the selected group address range opens. Enter the new name and press the **enter** key.

Delete group address range:

Select in the directory tree the group address range you wish to delete. Choose the menu item ***Edit*** and click ***Delete group address range***. Answer the confirmation prompt with ***Yes*** to delete the selected address range. The contained ranges and group addresses will be placed in the superordinate group address range. If there is no superordinate range, the group addresses will be deleted as well.

Delete group address range with group addresses:

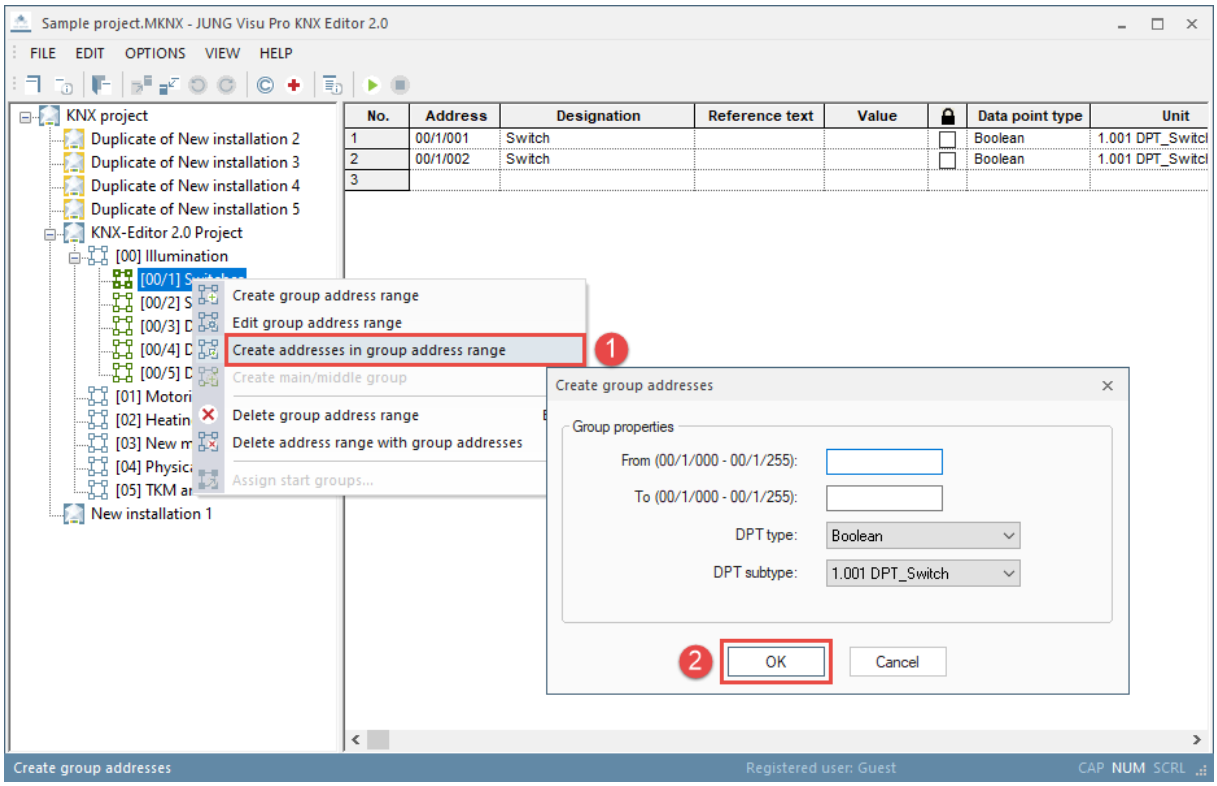
Select the group address range in the directory tree. Choose the menu item ***Edit*** and click ***Delete address range with group addresses***. Answer the confirmation prompt with ***Yes*** to delete the selected address range along with all ranges and group addresses contained in it.

4.5.4 Create Group Addresses

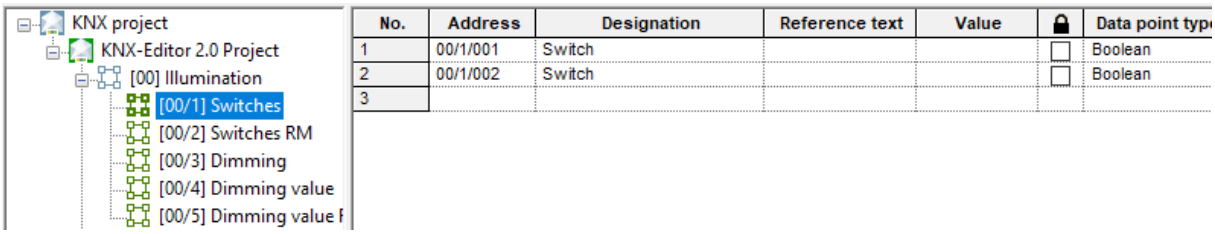
To create group addresses in a group address range,

- select the group address range, choose the menu item ***Edit*** and click ***Create addresses in group address range*** or
- right-click the group address range and click in the context menu the option ***Create addresses in group address range***.

The dialog ***Create group addresses*** opens.



The entry fields ***From*** and ***To*** allow to enter several group addresses in the selected address range, having the same data point type. If you already know the number of the required group addresses, you can create them all at once and configure them manually in the fields afterward. Here you can also change the middle group address.



4.5.5 Edit Group Addresses

Each group address requires further specifications to enable the KNX editor to properly interpret KNX telegrams:

Address: You can enter an address directly in the address column. Alternatively, select the group address range in the directory tree, right-click it and click in the context menu the option ***Create addresses in group address range***.

Designation:

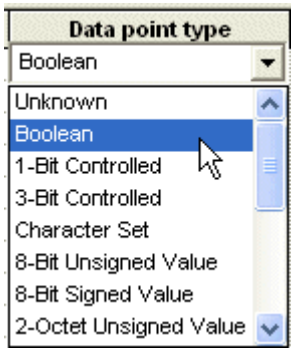
Enter a name for the group address. The designation serves the identification of the address during further processing of the data. This is a mandatory field! The KNX editor and other parts of the system will refer to the KNX group address by this name.

No.	Address	Designation
1	00/0001	Light 1

Value: If a group address receives values, they are displayed in this field. If write access is allowed, you may enter a value in this field and send a value to the KNX group address.

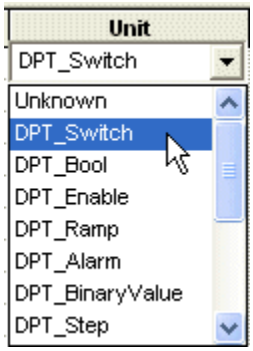
Reference text: An alternative text for the installation name which will be exported to the csv file as well.

Lock address: When this option is enabled, the data point will not be updated during an ETS import.



Data point type:

Type of the KNX address. To assign a data point type (DPT), click in the input field and select the desired data point type.



Unit:

After a type is set, an applicable KNX unit can be chosen. The specification of the unit is **optional**. If the unit has been defined for a group address, it will be shown in the messages list in a plain text message.

To assign a unit, click in the input field and select a unit from the dropdown list. The list automatically contains only the units which can be assigned to the data point type.

Rights:

To execute the desired action via the bus, the read and write rights must be set.

None: This is the default: Neither read nor write access is allowed. It is not possible to read the KNX group address nor to write values to it. However, spontaneous messages from this KNX group address can be received and interpreted.

Read: It is only possible to send read commands to the group address (no write commands).

Write: It is only possible to send a value to the KNX group address. The KNX editor denies read access to the address (important for e.g. switch objects).

Read/Write: It is possible to send a value to the group address and to read it.

Initialization: Here you can assign the group address to a start group.

Description: Enter a short description of the address (optional).

Ref.Unit (0): The notation of a certain state can be changed here (for example if an "off" instead of 0 is expected). This will be exported with the report data.

Ref.Unit (1): The notation of a certain state can be changed here (for example if an "on" instead of 1 is expected). This will be exported with the report data.

Note: *When you move the cursor to another row, either with the mouse or keyboard keys, the input of the current KNX group address will be committed. Leaving the edited row will commit the row's data!*

A new empty row will be displayed to enter the next KNX group address.

The cursor can be moved either with mouse click on a cell or keyboard arrow keys. The field under the cursor can be edited. To move the cursor between the cells of one row, use the <tab> and <shift>+<tab> keyboard keys.

Save entries:

Before the new group address definitions can be used, the data must be saved. The editor can only interpret telegrams from the KNX bus after the newly entered data have been saved.

To save the project, choose the menu item ***File*** and click ***Save*** or click the symbol in the toolbar.

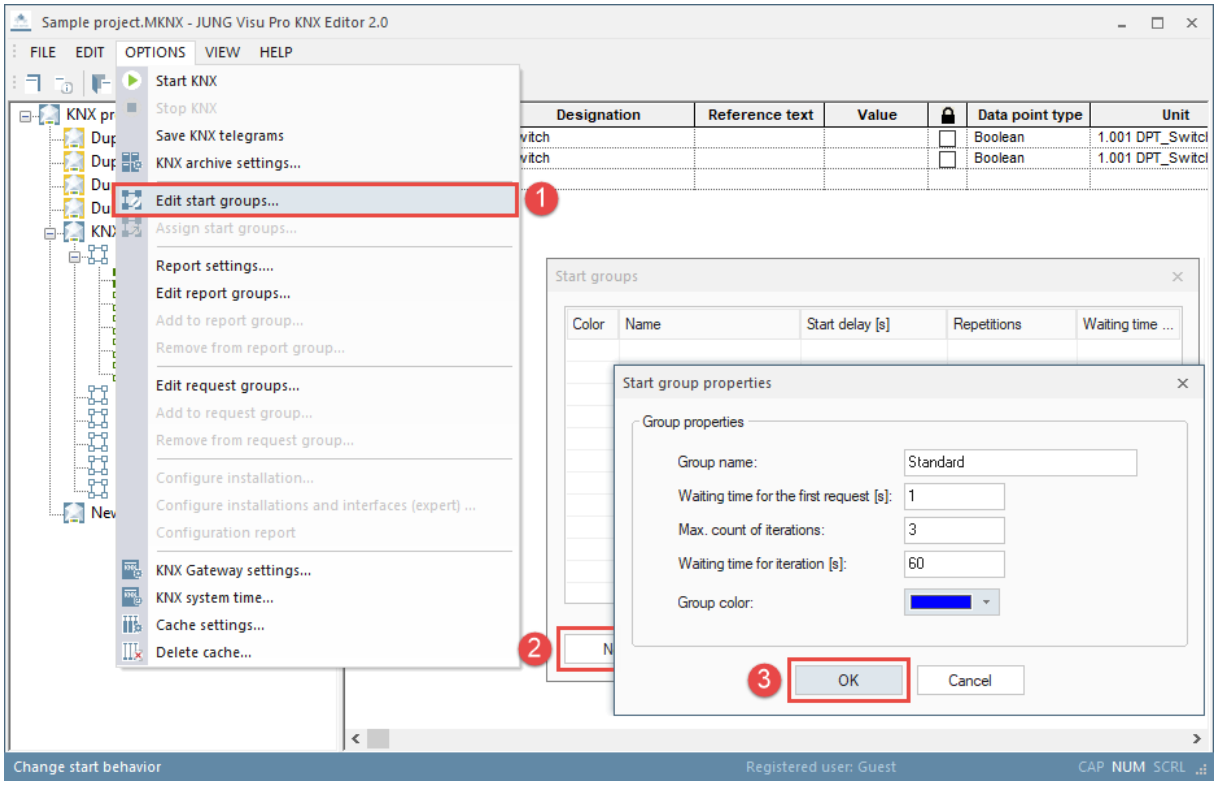
Warning: *Only after saving the changes will be interpreted by the bus!*

4.6 Start Groups

Through the **start groups**, group addresses can be specifically initialized. You can enter a waiting time to specify when a value should be initialized after the start of the project.
It is recommended to use this feature only for status variables.

Choose the menu item ***Options*** and click ***Edit start groups***. The dialog ***Start groups*** opens.

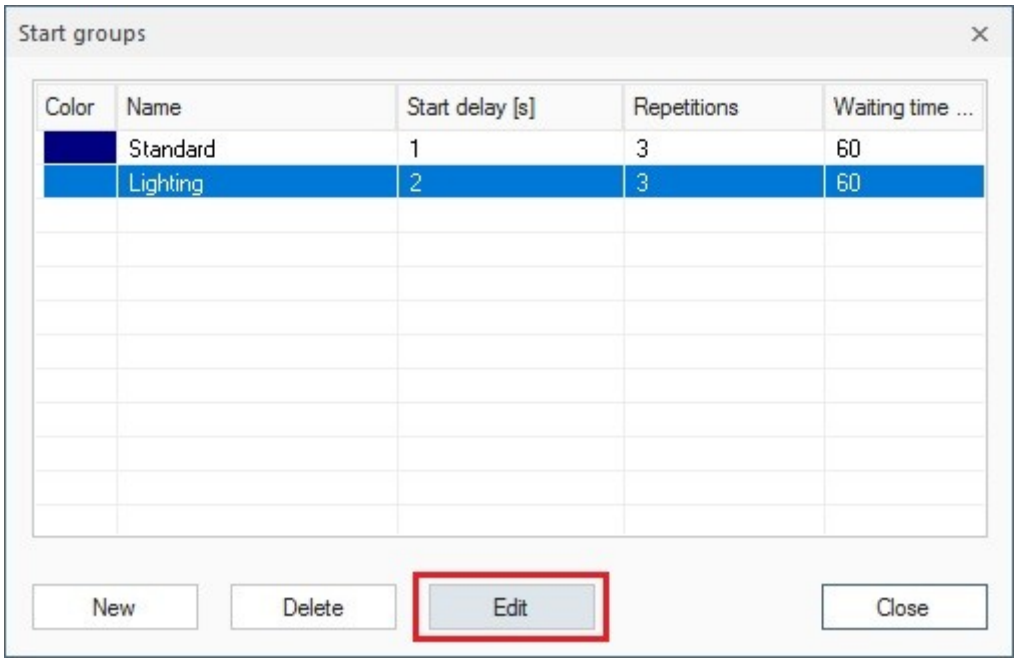
Create start group:



Click the button ***New***. The dialog ***Start group properties*** opens. Enter the settings and press the button ***OK*** to create the start group and to close the dialog. The newly created start group is now available in the dialog ***Start groups***.

Edit start group:

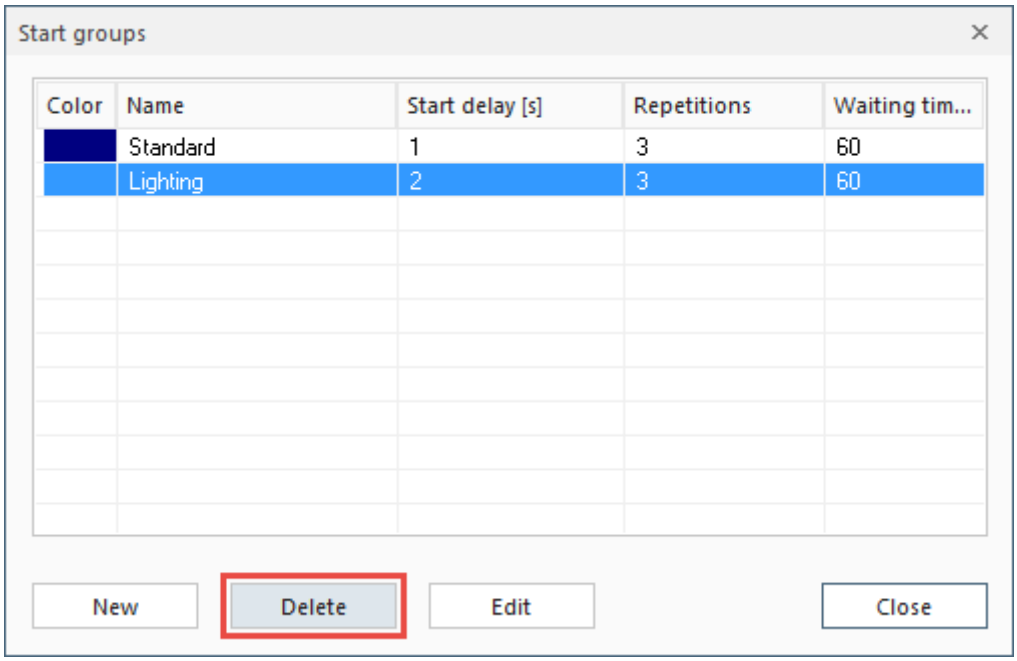
To edit the start group properties, select the start group and click the button ***Edit***. The dialog ***Start group properties*** opens.



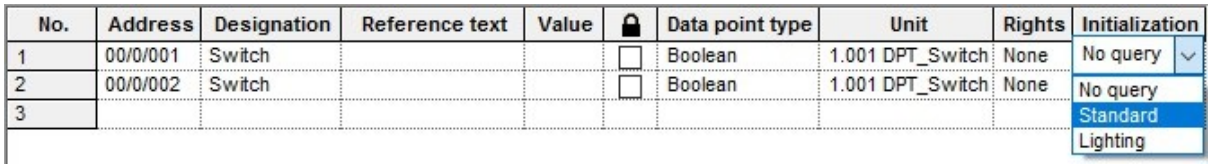
Edit the start group properties and click the button ***OK*** to commit the entries.

Delete start group:

To delete a start group, select the start group and click the button ***Delete***. Then click the button ***Close*** to quit the dialog ***Start groups***.

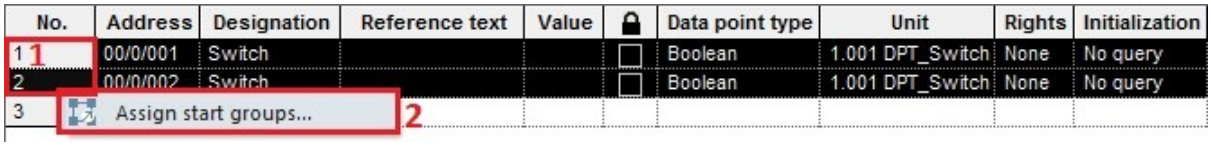


Assign start group:

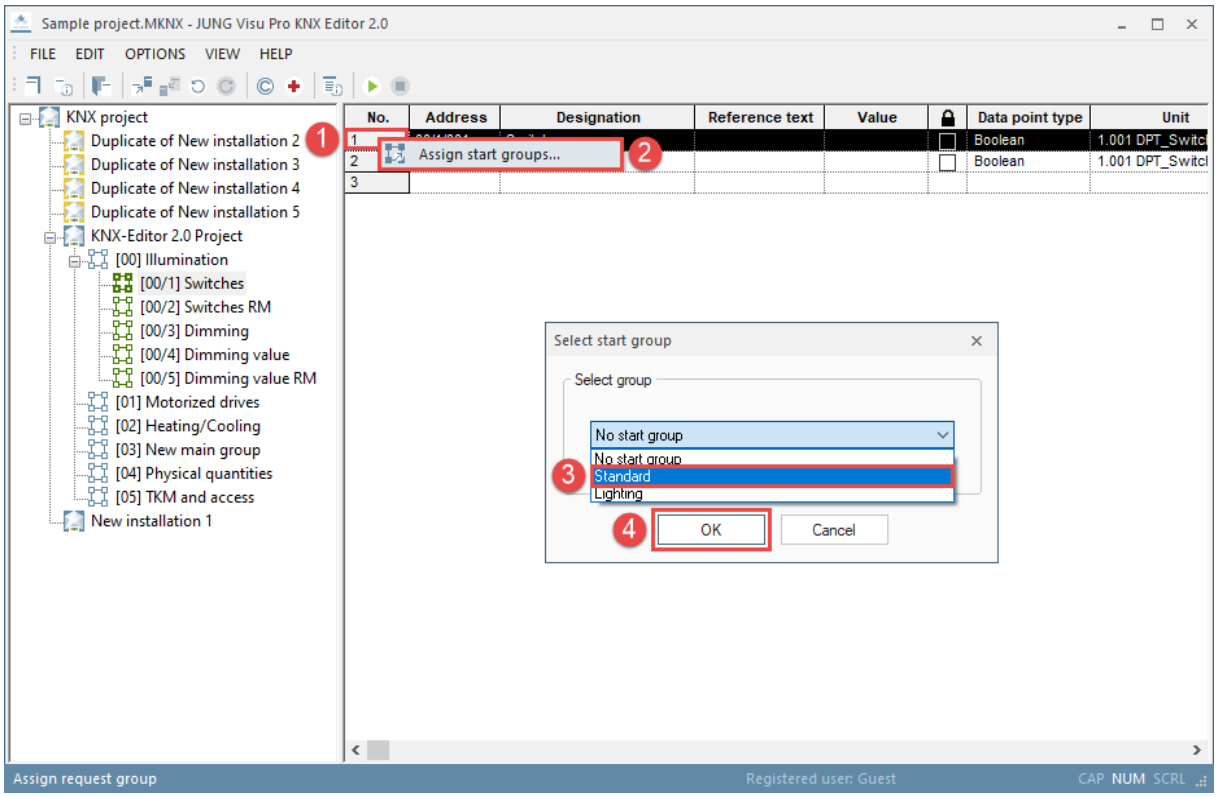


To assign a start group to a group address, click in the column ***Initialization***. Then select the start group in the combo box.

You can assign multiple group addresses to the same start group: Select the desired group addresses while pressing the control key in the first column of the schedule. Right-click to open the context menu and choose ***Assign start groups...*** The dialog ***Select start group*** opens.



Or: To assign all group addresses to a start group, click on the top left cell of the grid to select the whole table. Then right-click to open the context menu and click ***Assign start groups***. The dialog ***Select start group*** opens.



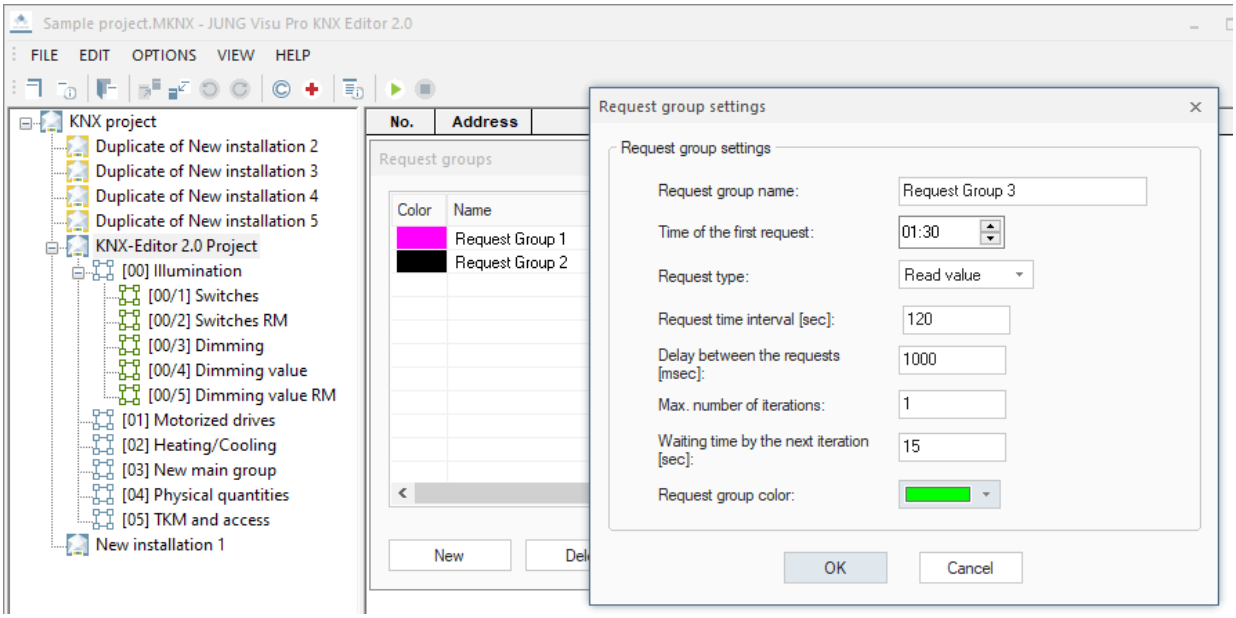
Select the desired start group and click ***OK*** to commit the entry.
All group addresses have now been assigned the start group ***Standard***.

4.7 Request Groups

Through the **request groups**, values can be read frequently. You can enter periodically when a value should be read or you can send 1 or 0 (if the end device needs a request value before sending back its value).

Click under ***Options*** at ***Edit request groups*** to open the dialog ***Request groups***.

Create request group:



Click at ***New*** to open the dialog ***Request group settings***. Insert the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Request groups.***

Request group name: Name of the group with which the group is listed.

Time of the first request: Time between 00:00 and 23:59.

Request type: Read value / Send 1 / Send 0.

Request time interval [sec]: Time specification between 10 and 86400 seconds.

Delay between the requests [msec]: Time specification must be a value between 250 and 60000 milliseconds. This option determines the break between different request groups (necessary at a project with many data points to not overcharge the bus and lose protocols).

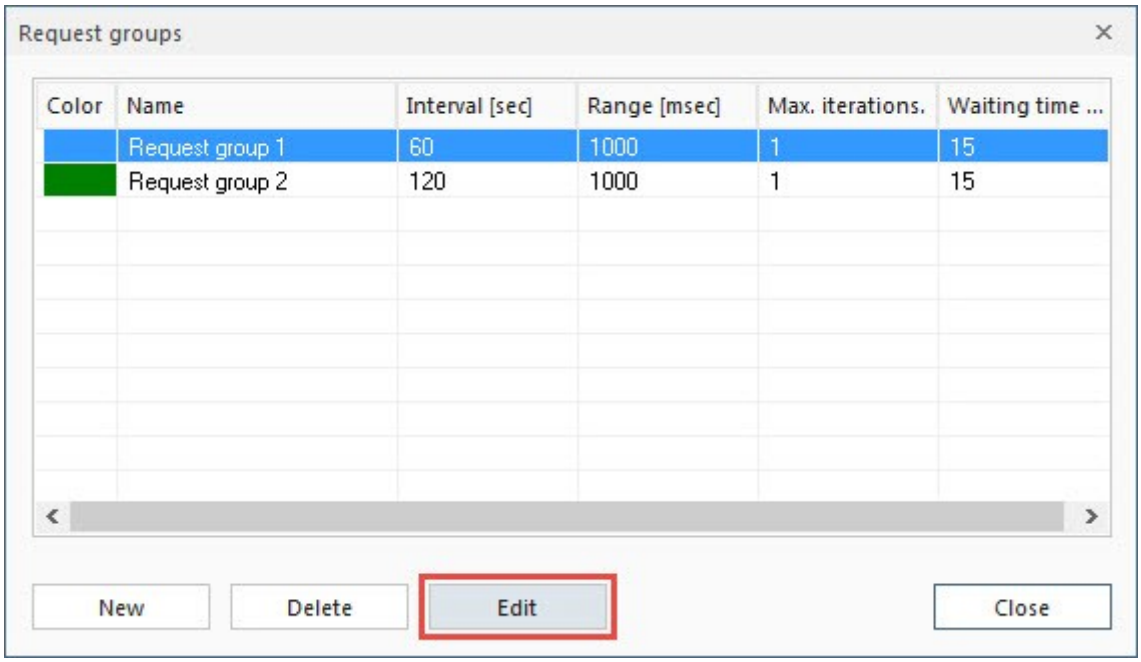
Max. number of iterations: Determines the number of repeats if the request failed.

Waiting time by the next iteration[sec]: Determines the time between the repeats.

Request group color: Set a color to have a better discrimination of the groups.

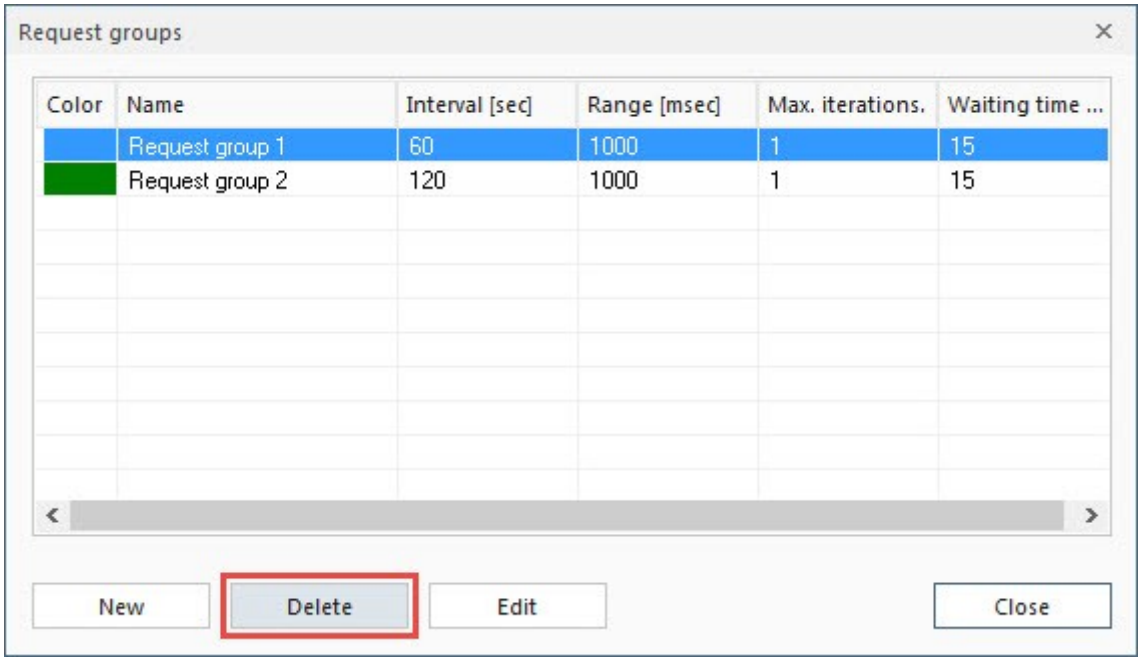
Edit request group:

To edit a request group, select the request group and click the button ***Edit***. The dialog ***Request group settings*** opens.



Edit the group settings and save the changes by clicking ***OK***.

Delete request group:



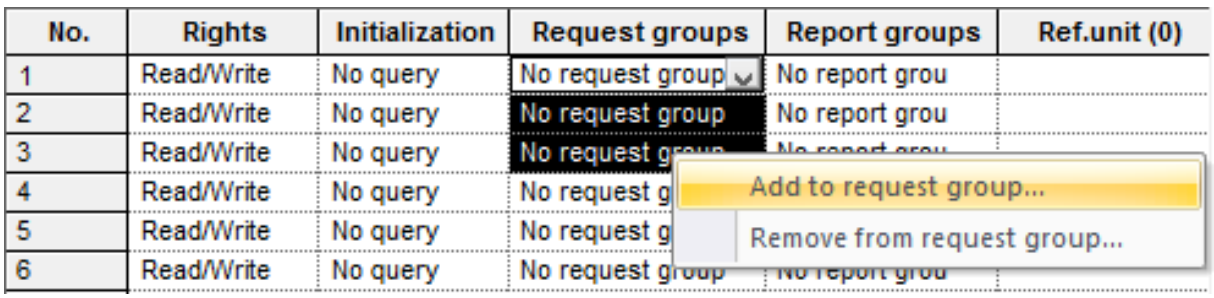
To delete a request group, select the request group and click the button ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Request groups***.

Assign request group:

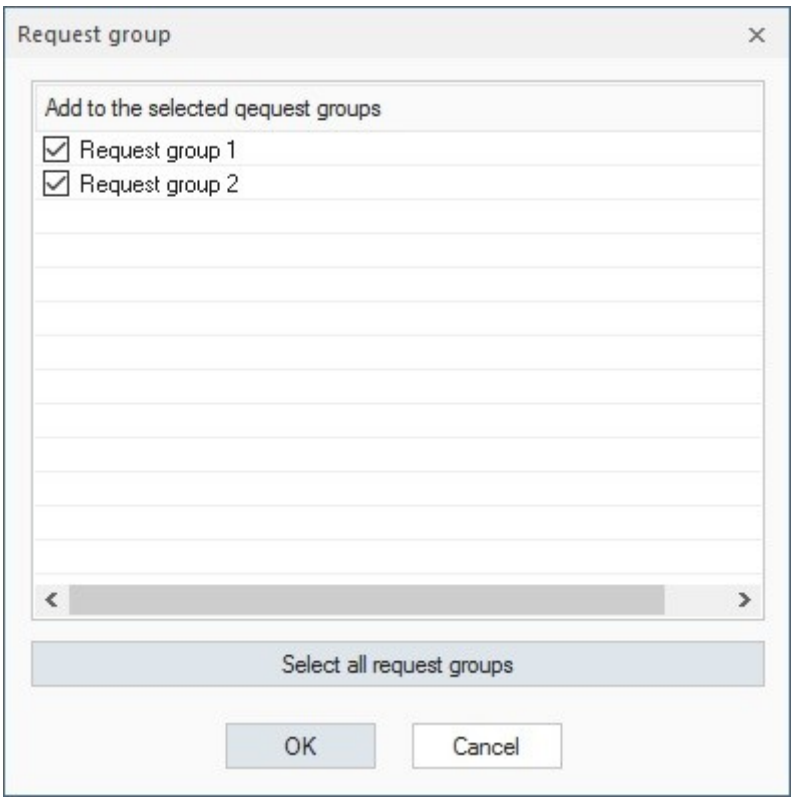
No.	Unit	Rights	Initialization	Request groups
1	1.001 DPT_Switch	Read/Write	No query	No request group
2	1.001 DPT_Switch	Read/Write	No query	Request group 1
3	1.001 DPT_Switch	Read/Write	No query	Request group 2

To assign a request group to a group address, click in the column ***Request groups*** to select one or multiple request groups.

Or: To assign a selection of group addresses to a request group, select the first group address in the column ***Request groups*** and draw the mouse over the beneath lying lines. Right-click on the selected elements to open the context menu and choose ***Add to request group***.



The dialog ***Request group*** opens.



Set a checkmark at the desired request groups and confirm with ***OK***.

No.	Unit	Rights	Initialization	Request groups
1	1.001 DPT_Switch	Read/Write	No query	2 request groups
2	1.001 DPT_Switch	Read/Write	No query	2 request groups
3	1.001 DPT_Switch	Read/Write	No query	Request group 2
4	1.001 DPT_Switch	Read/Write	No query	Request group 1
5	1.001 DPT_Switch	Read/Write	No query	Request group 1
6	1.001 DPT_Switch	Read/Write	No query	Request group 2

For every group address is the request group now visible or, if a group address has multiple request groups, the number of request groups.

Delete request group:

Deleting request groups is analog to assigning request groups.

4.8 Report Groups

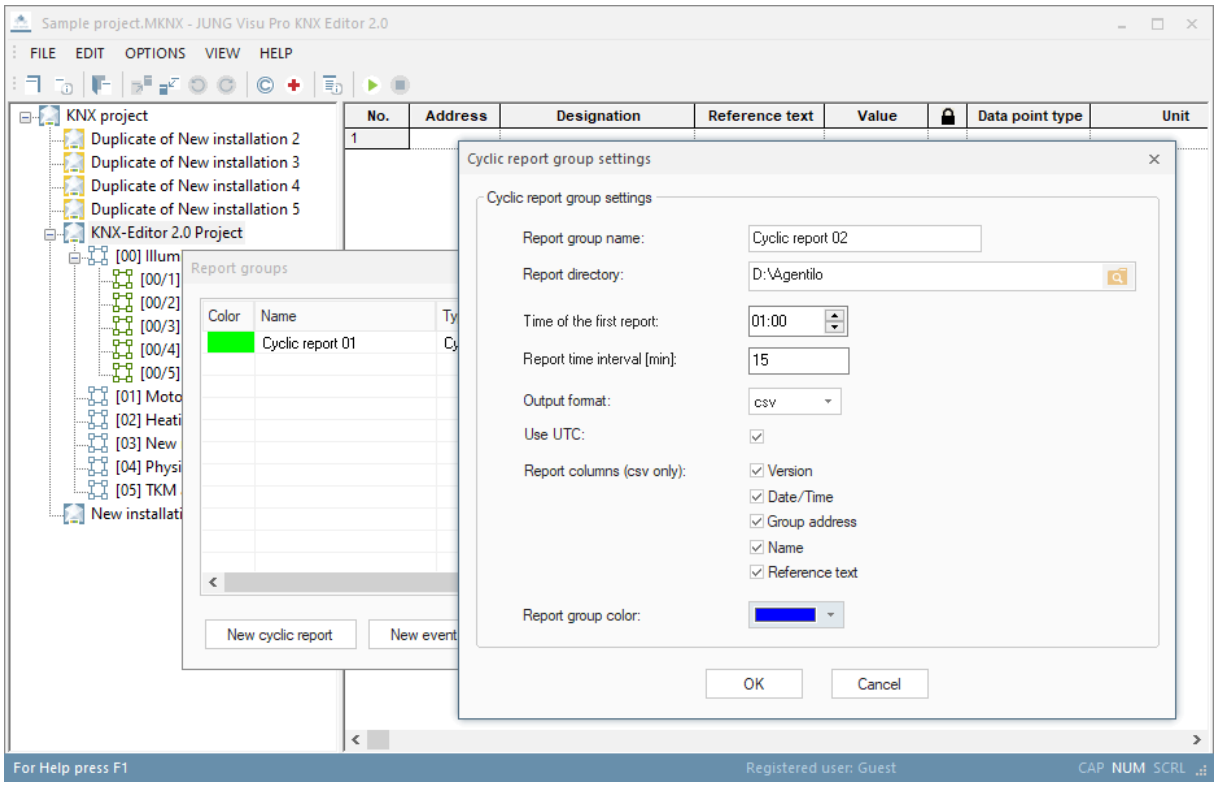
The report groups divide into: ***cyclic reports*** and ***event reports***.

Cyclic reports record from a certain time, in defined periods, the state of the data points within the group.

Event reports check after a preset time period if the value within the group changed and record this value in an event report. It can additionally be determined after what a time period of constant value the value should be recorded as an event.

Click under ***Options*** at ***Edit report groups*** to open the dialog ***Report groups***.

Create report groups:



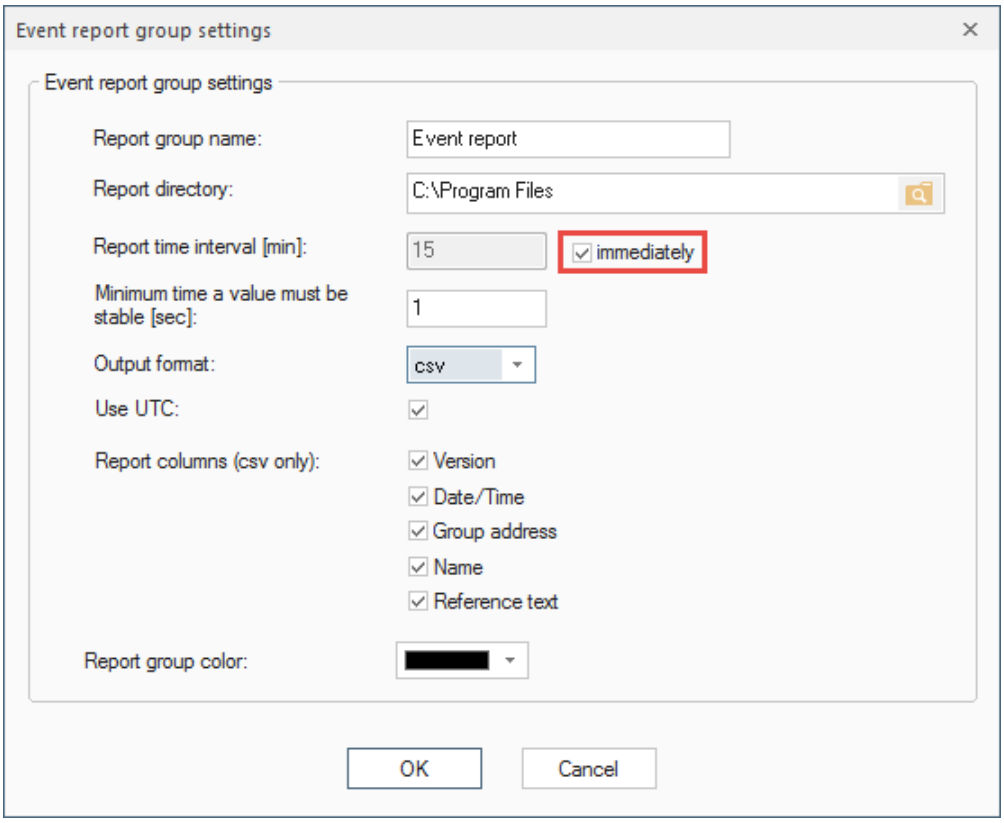
Click either at the button ***New cyclic report*** or ***New event report*** to open the dialog ***Report group – settings***. Enter the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Report groups***.

Report group name: Name of the group with which the group will be listed.

Report directory: The path to the folder that contains the saved reports.

Time of the first report (only at cyclic report): Time between 00:00 and 23:59.

Report time interval [min]: The time in minutes after which a new report is written. Oriented at the time of the first report.
Cyclic report: Must be between 1 and 1440 minutes, 1440 means a daily interval.
Event report: Must be between 0 and 1440 minutes, 1440 means a daily interval. If you set a check mark at ***immediately***, a report is created as soon as the event is detected. Alternatively you can enter a 0 at ***Report time interval***.



Minimum time a value must be stable [sec] (only at event report): Between 1 and 60 seconds.

Output format: The reports are written either into a xml or a csv file.

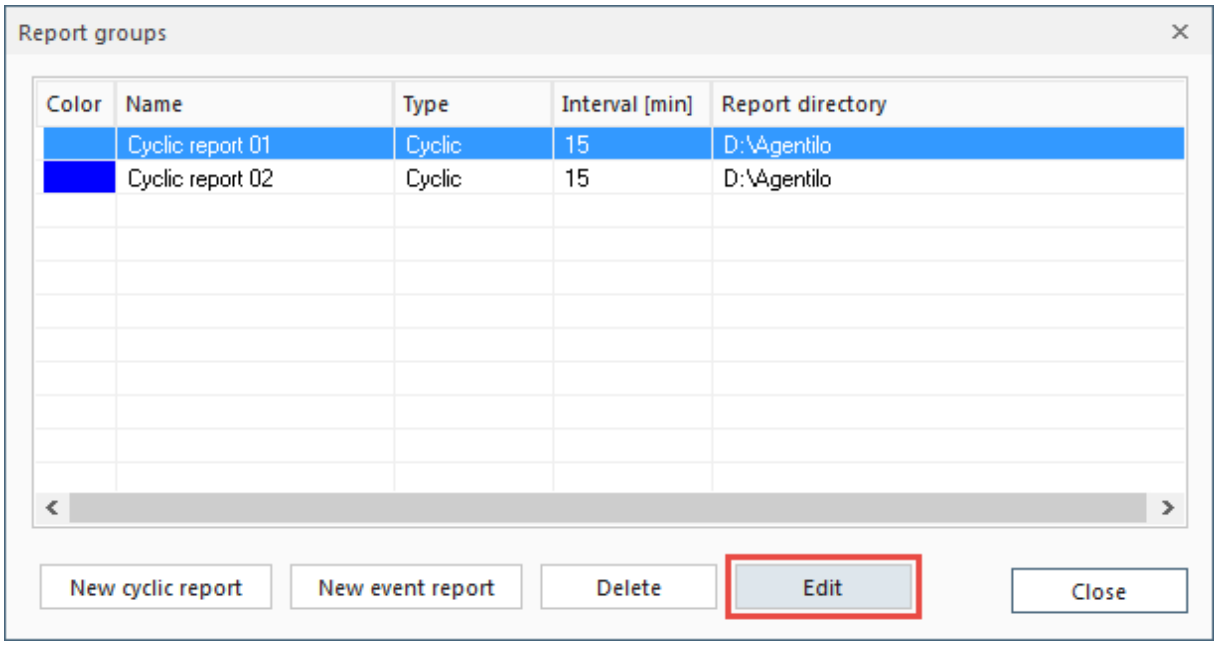
Use UTC: Activates or deactivates the Unit-Time-Code for writing the report.

Report columns (csv only): Defines which additional information should be written into the report.

Report group color: For a better discrimination of the different groups, a color can be set.

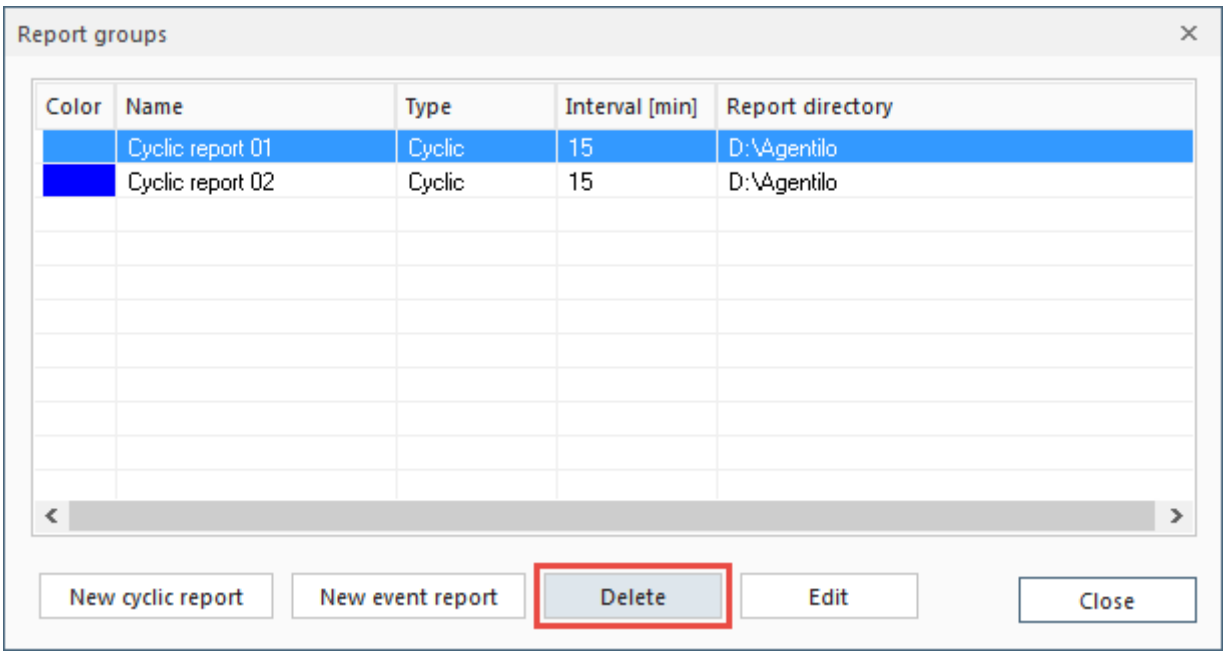
Edit report group:

To edit a report group, select the report group and click at ***Edit***. The dialog ***Report group settings*** opens.



Edit the group settings and confirm the changes with ***OK***.

Delete report group:



To delete a report group, select the report group and click at ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Report groups***.

Assign report group:

No.	Rights	Initialization	Request groups	Report groups
1	Read/Write	No query	No request group	No report grou
2	Read/Write	No query	No request group	Cyclic 01
3	Read/Write	No query	No request group	Event 01
4	Read/Write	No query	No request group	Cyclic 02

To assign a report group to a group address, click at the column ***Report groups*** to select a report group.

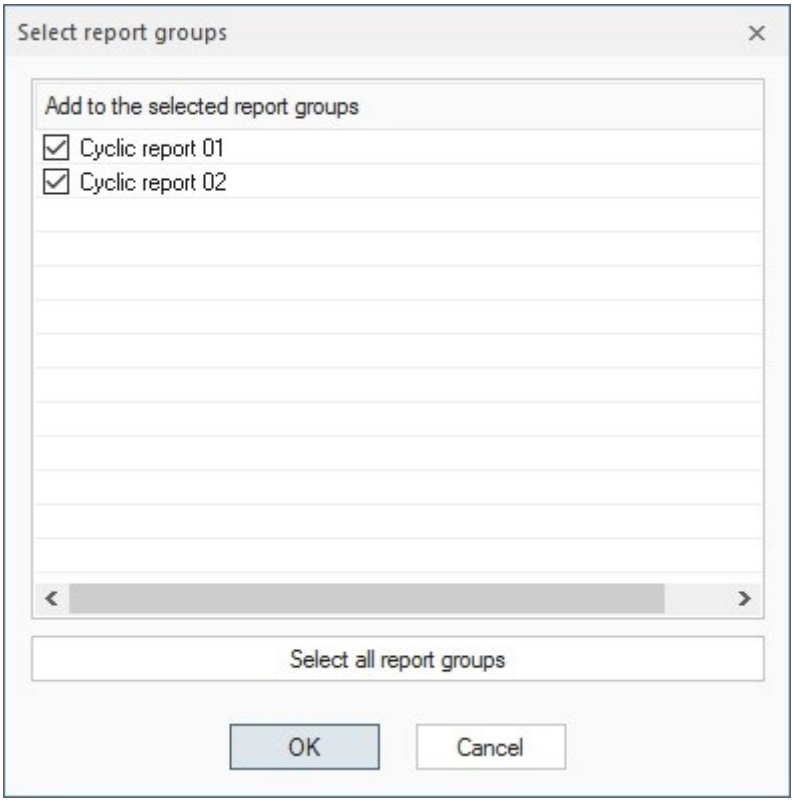
Or: To assign a selection of group addresses to a report group, select the first group address in the column ***Report groups*** and draw the mouse over the beneath lying lines. Right-click at the selected elements to open the context menu and click at ***Add to report group***.

No.	Initialization	Request groups	Report groups	Ref.unit (0)	Ref.unit (
1	No query	No request group	No report grou		
2	No query	No request group	No report grou		
3	No query	No request group	No rep		
4	No query	No request group	No rep		
5	No query	No request group	No rep		

Add to report group...

Remove from report group...

The dialog ***Report group*** opens.



Set a checkmark at the desired report groups and confirm the selection with ***OK***.

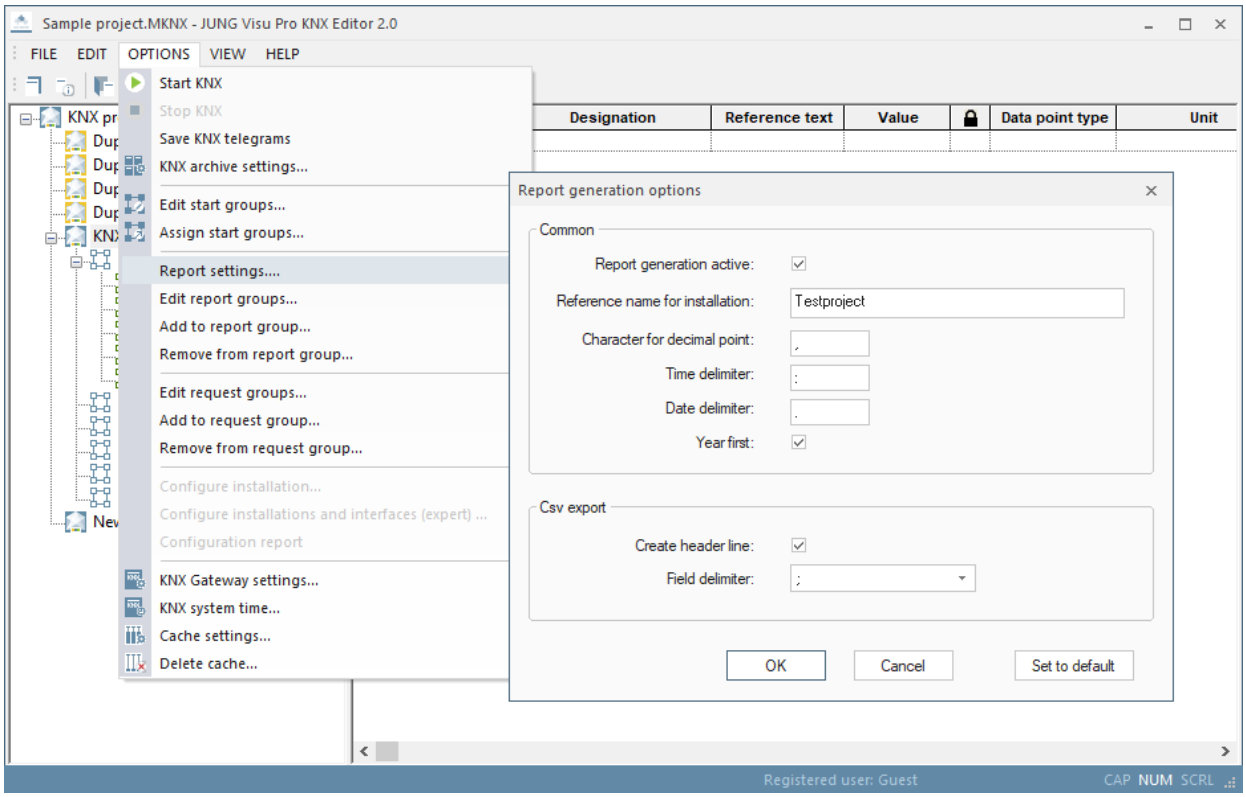
No.	Rights	Initialization	Request groups	Report groups
1	Read/Write	No query	No request group	3 report group
2	Read/Write	No query	No request group	3 report group
3	Read/Write	No query	No request group	Cyclic report 0
4	Read/Write	No query	No request group	Event report 01
5	Read/Write	No query	No request group	Cyclic report 0
6	Read/Write	No query	No request group	3 report group
7	Read/Write	No query	No request group	2 report group
8	Read/Write	No query	No request group	2 report group

For every group address is the report group now visible or, if a group address has multiple report groups, the number of report groups.

Delete report groups:
Deleting report groups is analog to assigning report groups.

4.8.1 Report Formatting

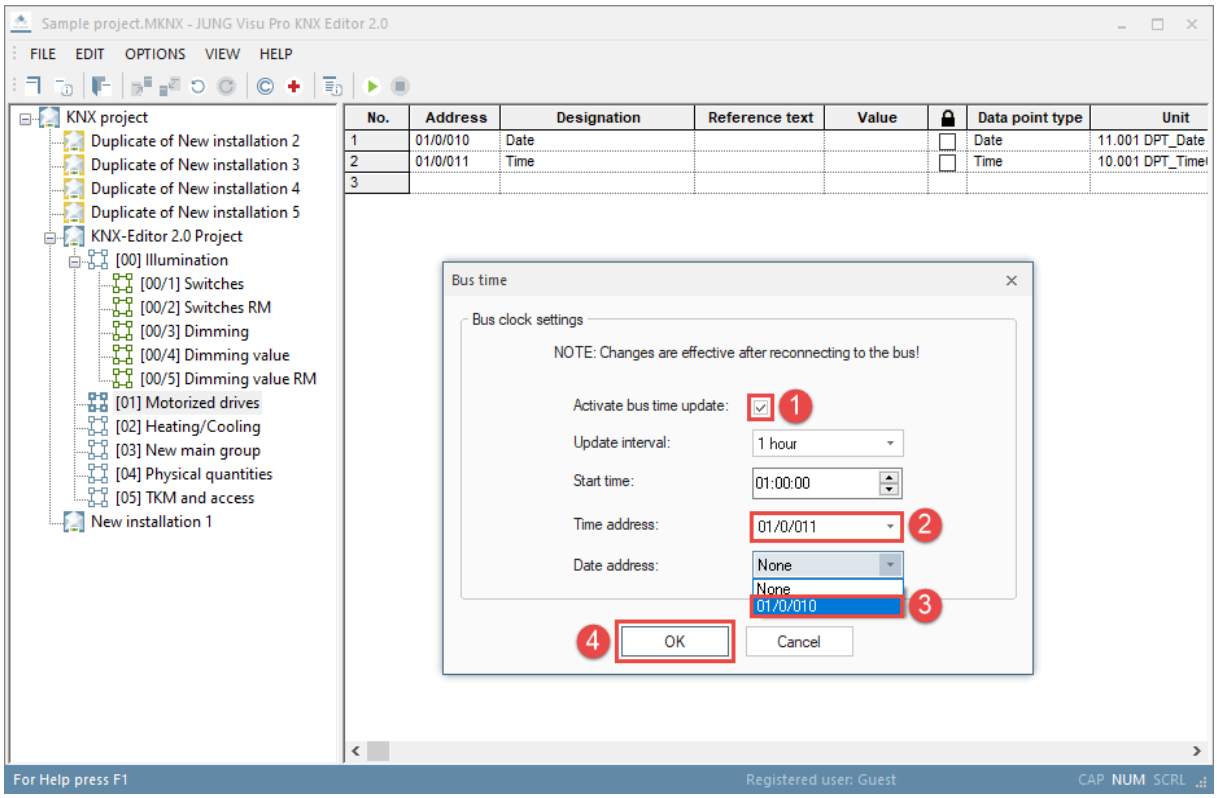
If the produced csv format is not of use for further editing, it is possible to change the formatting of the reports. Click under ***Options*** at ***Report settings*** to open the dialog ***Options for the report creation***.



Change the settings and confirm with ***OK***.

4.9 KNX System Time

To activate the actualization of the bus time, choose the menu item ***Options*** and click ***KNX system time***. The dialog ***Bus time*** opens.



To **actualize** the bus time **automatically**, activate the check box ***Activate bus time update***.

Set the update interval and the start time and choose the write addresses for date and time. Press ***OK*** to commit the changes and to close the dialog.

4.10 Synchronize PC Time with KNX Time

By synchronizing bus and PC time, the difference between bus and PC time shall be minimized.

The synchronization mechanism is not a method for setting a specific time or date on a PC via bus telegrams. In particular, it is also not intended to perform the summer/winter time changeover on a PC.

The synchronization is only possible within a certain time slot. The time information from the bus is evaluated by the KNX editor between 5:00 and 22:00. The synchronization will be done if the difference between bus and PC time is greater than or equal to 5 seconds.

To do so the KNX editor evaluates 2 bus telegrams: One telegram containing the current date (DPT 11.001) and one telegram containing the current time (DPT 10.001). The KNX editor takes the bus time into account only if the interval between these two telegrams is less than 10 seconds.

A suitable KNX device (e.g. weather station) must be used and configured for the cyclical updating of the bus time.
Possible update intervals are:

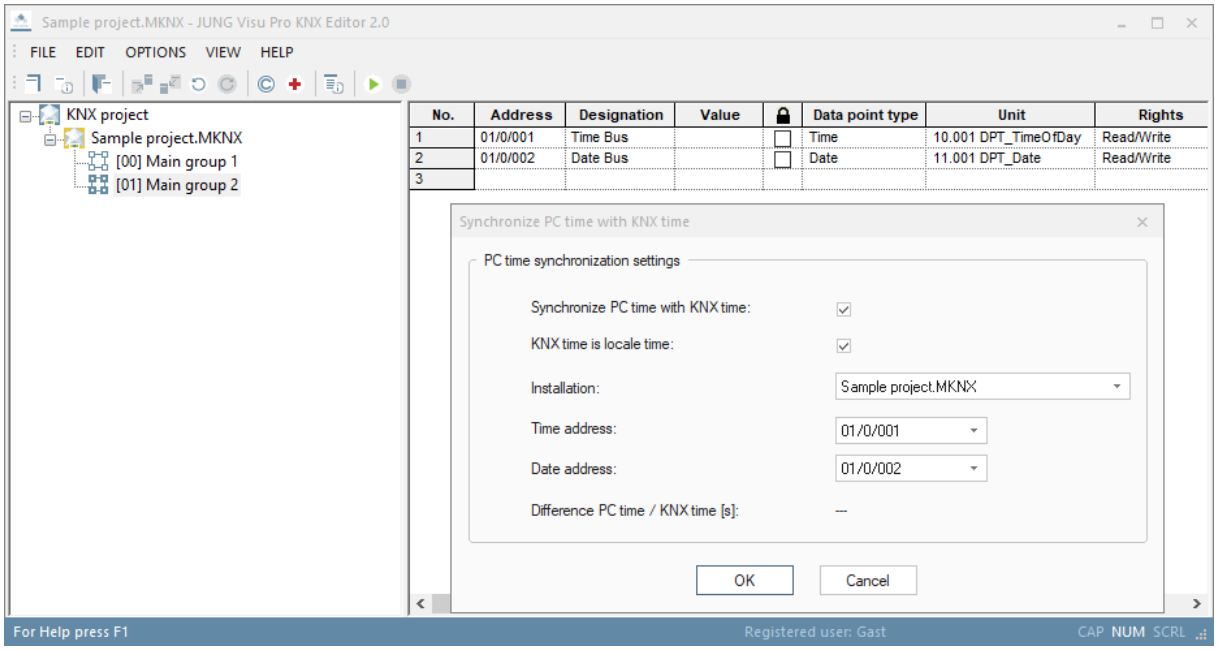
- Hourly transmission of the bus time
- Daily transmission of the bus time.
  If the bus time is transmitted once a day, the transmission time must be within the time slot described above, e.g. at 6:00 a.m.

Depending on the KNX device, the transmission of the current time can also be triggered via a group address. In this case, a corresponding Request Group can be created in the KNX editor, which queries the bus time every hour or in 24 hour intervals.

Recommendation:

- Daily sending of the time and date between 05:00 and 22:00.

To synchronize the PC time with the KNX time, click on ***Options*** in the menu and select ***Synchronize PC time with KNX time***… Following dialog will be opened:



To **automatically synchronize** the PC time you have to enable the checkbox ***Synchronize PC time with KNX time***.

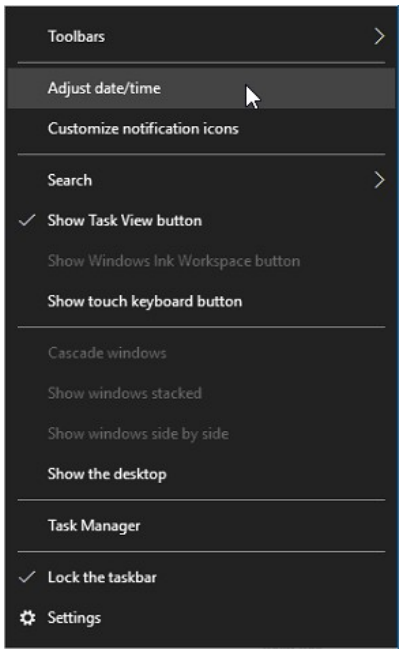
If the KNX time does not match the locale time zone, but UTC instead, you have to disable the checkbox ***KNX time is locale time***.

Choose an installation.

Choose the read addresses for time and date.

Press ***OK*** to commit the changes and to close the dialog.

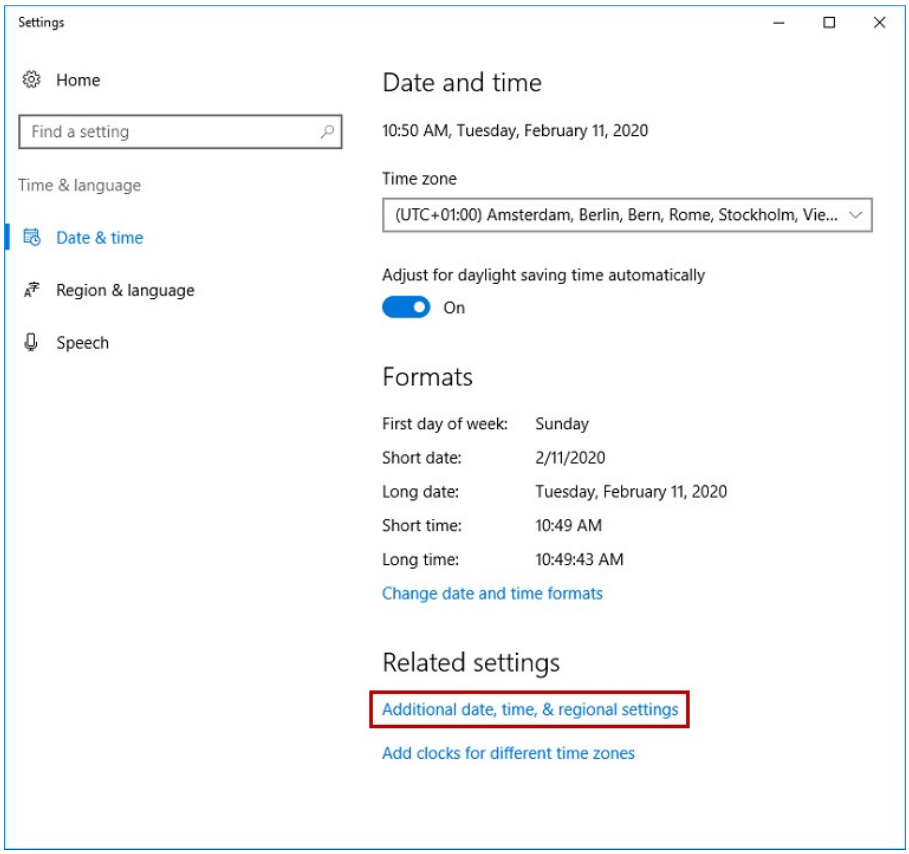
To prevent Windows 10 from constantly updating the time, Internet synchronization must be disabled in the time settings:



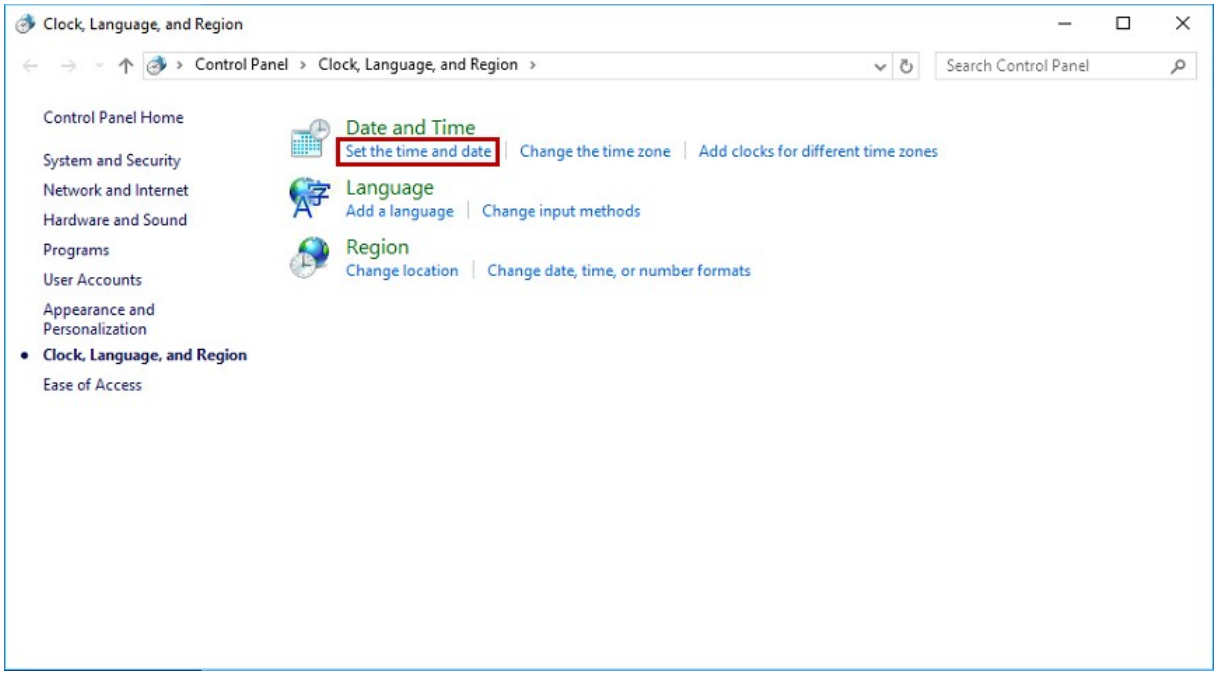
click on the clock with the right mouse button (at bottom on the right on the screen)

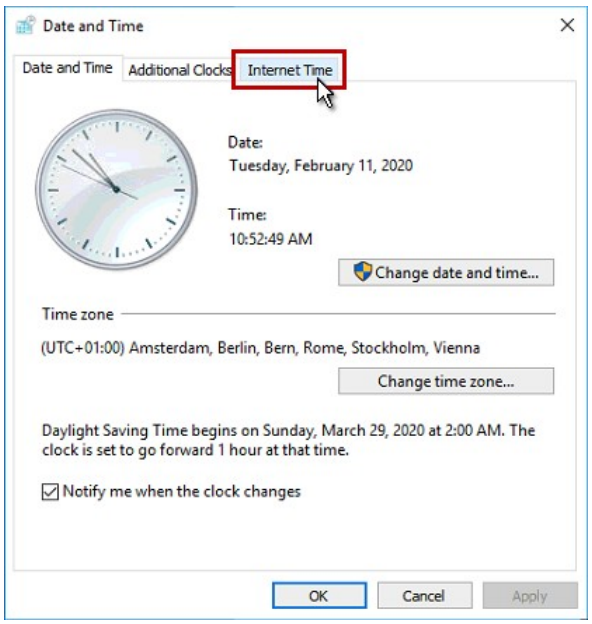
select ***Adjust date/time***

under *Related settings*, click on ***Additional date, time, & regional settings***

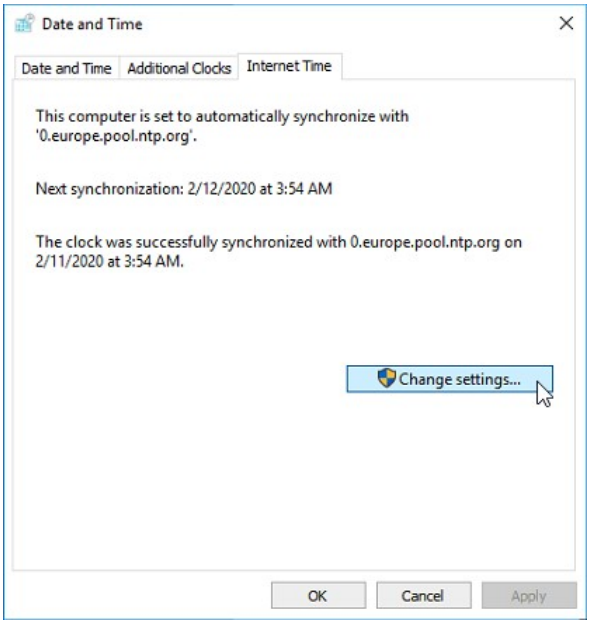


then click on ***Set the time and date***

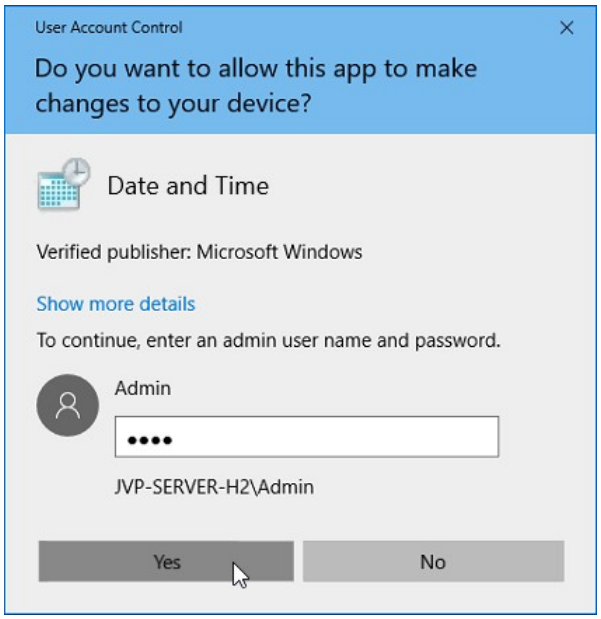




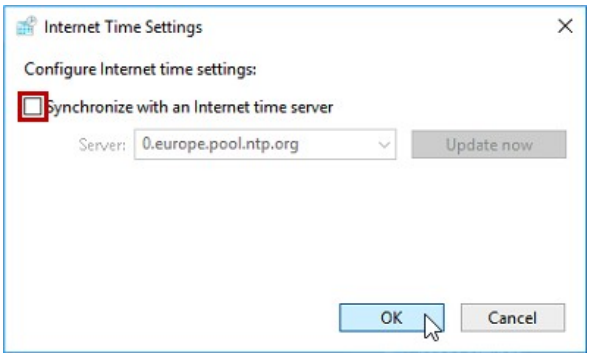
then select ***Internet Time***



click on ***Change settings...***



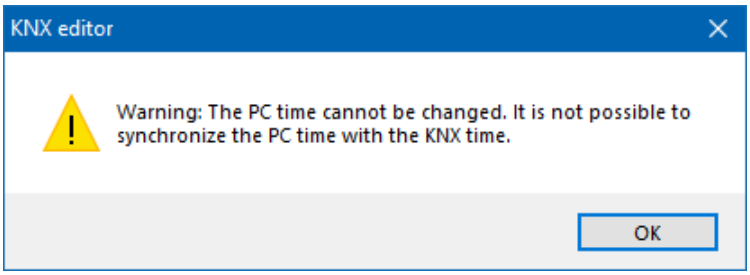
Enter the password of the administrator, click ***Yes***



Remove the check mark for ***Synchronize with an Internet time server***, confirm with ***OK***

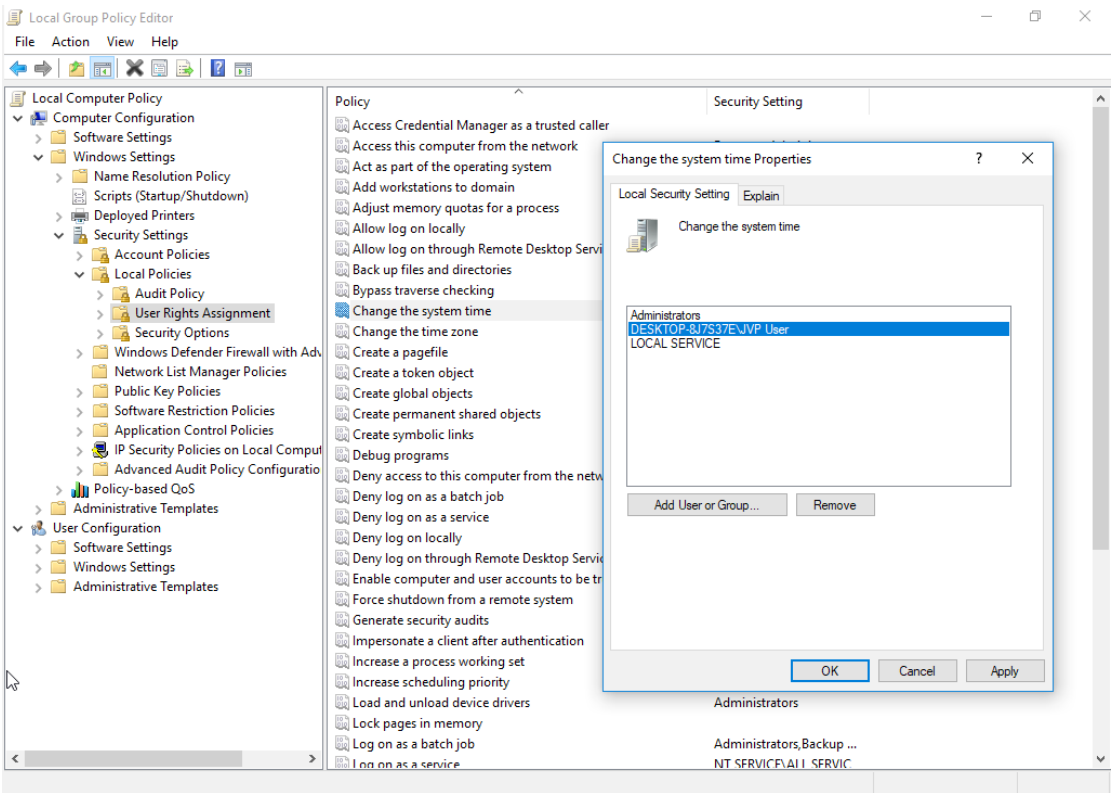
Warning:

To change the PC time, the KNX-Editor needs Administrator rights! A dialog will be shown, if you click ***Synchronize PC time with KNX time*** *and do not have the needed rights.*



To open the KNX-Editor with administrator rights, right click on JUNG VISU Pro and choose ***Run as administrator****.*

Alternatively, you can grant your normal Windows user the necessary rights to change the PC time. This is necessary if the editor starts automatically.
To do this you need to configure the local group policies, which are ***not available*** *for home editions of Windows.*
To make changes in the 'Local Group Policy Editor', ***it is necessary to change the user and log in as 'Admin'.*** *From the list choose Computer Configuration → Windows Settings → Security settings → Local Policies → User Rights Assignment and choose* ***Change the system time****. A dialog opens, which allows you to add your current user for this policy. Restart your system after you added the user.*



4.11 Application of KNX Data Types

The KNX data point types specify the meaning of transferred bus data. There are different interpretations of the raw data depending on their data type.

The data packet **cf c7**, received by the KNX editor, can have completely different meanings:

-291.84,	interpreted as 2 Octet Float Value
53191,	interpreted as 2 Octet Unsigned Value
–12345,	interpreted as 2 Octet Signed Value

Attention: *The correct assignment of KNX data point types to group addresses is a fundamental precondition for all applications which use the KNX editor as process connection!*

The following table gives a short introduction to the meaning of KNX data point types:

Boolean	**Unit** **1.001 –1.022** **DPT**	**1 bit** data length, formerly EIS1. This data type is used to transmit switch states (e.g. 0/1, on/off, open/closed, up/down, …). This type is particularly used to switch devices on or off or to receive device states (lamps, dimmers, shutters, relay, …).
1-Bit Controlled	**Unit** **2.001 – 2.012** **DPT**	**2 bit** data length, formerly EIS8. This data type is used for priority-controlled switch operations.
3-Bit Controlled	**Unit** **3.007 – 3.008** **DPT**	**4 bit** data length, formerly EIS2, subtype control. This data type is used to increase or decrease the set value in steps or stop the movement (e.g. to control a dimmer).
Character Set	**Unit** **4.001- 4.002** **DPT**	**1 byte** data length, formerly EIS13. This data type is used to transfer a character over the bus. ***Attention:*** *The visualization does not yet support this data type.*

8 Bit Unsigned Value	**Unit 5.001, 5.003, 5.004, 5.010**	**1 byte** data length, formerly EIS6 or EIS14. This data type is used to transfer 8 bit unsigned integer values (e.g. counter values).

> ***Attention:*** *Normally the range of the 8 bit value is from 0 to 255, but the interpretation can differ depending on the subtype selection. If the subtype* scaling *is selected, the value is interpreted as a percent value in the range from 0 to 100%. This subtype can be used to set the values of dimming actuators. If the subtype* wind direction *is selected, the value is interpreted as an angle in the range from 0 to 360°.*

Subtype 5.001 DPT_scaling	The value is interpreted as a percent value in the range from 0 … 100%.
Subtype 5.003 DPT_angle	The value is interpreted as an angle in the range from 0 … 360°.
Subtype 5.010 DPT_Value_Ucount	The value is interpreted in the range from 0 … 255, e.g. as counter value.

8 Bit Signed Value	**Unit 6.010 DPT**	**1 byte** data length, formerly EIS14, used to transfer an 8 bit signed integer value in the range from –128 to +127.
2 Octet Unsigned Value	**Unit 7.001, 7.010, 7.600 DPT**	**2 byte** data length, formerly EIS10, used to transfer a 2 byte unsigned integer value (e.g. a counter value).
2 Octet Signed Value	**Unit 8.001 DPT**	**2 byte** data length, formerly EIS10 signed, used to transfer a 2 byte signed integer value in the range from – 32768 to +32767.
2 Octet Float Value	**Unit 9.001-9.006, 9.010-9.011. 9.020-9.021 DPT**	**2 byte** data length, formerly EIS5, used to transfer a 2 byte analog value (e.g. values from a temperature sensor).
Time	**Unit 10.001 DPT**	**3 byte** data length, formerly EIS3, used to transfer a time information (e.g. from a KNX synchronization clock).
Date	**Unit 11.001 DPT**	**3 byte** data length, formerly EIS4, used to transfer a date value.

4 Octet Unsigned Value	**Unit 12.001**	**4 byte** data length, formerly EIS11, used to transfer a 4 byte unsigned integer value (e.g. a counter value).
4 Octet Signed Value	**Unit 13.001**	**4 byte** data length, formerly EIS11 signed, used to transfer a 4 byte signed integer value in the range from -2147483648 to +2147483647.
4 Octet Float Value	**Unit 14.001-14.079 DPT**	**4 byte** data length, formerly EIS9, used to transfer a 4 byte high precision analog value according to IEEE754 standard.
Access	**4 byte** data length, formerly EIS9, used for access control applications.	
String	**Unit 16.000-16.001 DPT**	**14 byte** data length, formerly EIS15, used to transfer strings.
UTF-8 String	**Unit 28.001 DPT**	Used to transfer strings in UTF-8 format.
Colour RGB	**Unit 232.600 DPT**	**3 Byte** data length, used to transfer colors in RGB format.
DateTime	**Unit 19.100 DPT**	**8 Byte** data length, used to transfer date and time.
Colour RGBW	**Unit 251.600 DPT**	**6 Byte** data length, used to transfer colors in RGBW format.

Attention:
The visualization does not yet support this data type.

4.12 ETS Import of KNX Group Addresses

In most cases it is more convenient to import KNX group addresses from ETS than to create them manually. The KNX editor is able to import KNX group addresses from **ETS2** versions 1.1, 1.2 and 1.3.

There are two methods:

a) Import the data from an ETS print report which has been redirected to a file. This method applies to **ETS2**, versions 1.1, 1.2 and 1.3.
b) Import the data from a file which has been created with the function OPC export. This method applies to **ETS2**, Version 1.3 and **ETS3**.

Data from **ETS4** and **ETS5** can be exported to a **project export** file which can be imported in the KNX editor.

> ***Important:*** *The KNX editor does not require ETS to be installed at the same PC. It is sufficient to copy the ETS output files to the PC.*

Please consider the following notes when importing group addresses from ETS:

- Assign **unique names** to the group addresses in the ETS project. It is important to assign meaningful names to the group addresses in order to facilitate the identification of addresses and plane text messages in the KNX editor and the visualization.

- Make sure to set the **data point types** already in the ETS project. The KNX editor is able to import the data point types from the ETS print file. If you have configured the data point types correctly in the ETS project, you do not have to enter them again in the KNX editor. This facilitates in particular the updated import of addresses after changes have been carried out in the ETS project.

- Edit the imported group addresses after the import to the KNX editor:

  → Check whether the **data point types** are set correctly;

  → Set the **read and write rights**. The read and write rights for the KNX editor (and for programs using the KNX editor) **cannot** be imported from ETS. They must be set manually.

4.12.1 Create ETS Group Addresses Print Report

Create ETS export file in ETS:

1. Open the ETS project in ***Project Design***.

2. Open the group address browser (Menu: ***Design***, ***Group Addresses***).

3. Choose the menu item ***Project*** > ***Page Setup*** and click ***Layout***. Click in the layout dialog ***reset the selection of Title page, Header, Footer***.

4. Mark the **root node** in the group address browser.

5. Choose the menu item ***Project*** and click ***Print***. In the print dialog click ***Print to File*** and click the button ***OK*** to start the export.

6. A print dialog opens. Choose the target directory, target file name, and select the extension ***.txt***.

7. To import the output file into the KNX editor, choose the menu item ***File*** and click ***Import ETS project***.

> ***Important:*** *With ETS the installed printer driver must support the function* **printing in a file***. As a remedy install the WINDOWS standard printer driver.*

4.12.2 Import OPC Export File from the ETS

The KNX editor can import group addresses from a file which has been created from ETS via the function ***OPC Export***.

Create OPC export file in ETS3:

1. Choose the menu item ***Extras*** and click ***Options***. Select the tab ***Presentation***. Select under ***Browser*** the check box ***Two Level Group Addresses***.

2. Choose the menu item ***File*** and click ***Extract Data (e.g. OPC)***. The dialog ***Foreign Format*** opens.

3. Click ***Export to OPC Server***. Specify the target directory and a file name for the OPC project file. Then click ***Export***. The default extension of the OPC project file is ***.esf***.

4. To import the file into the KNX editor, choose the menu item ***File*** and click ***Import ETS project***.

5. To toggle between the 2-level, 3-level and the free address structure, choose the menu item ***Edit*** and click the desired address structure.

Create OPC export file in ETS4 and ETS5:

1. Switch in ETS4 to the project overview and select the desired project. The button ***Export*** in the toolbar will be activated.

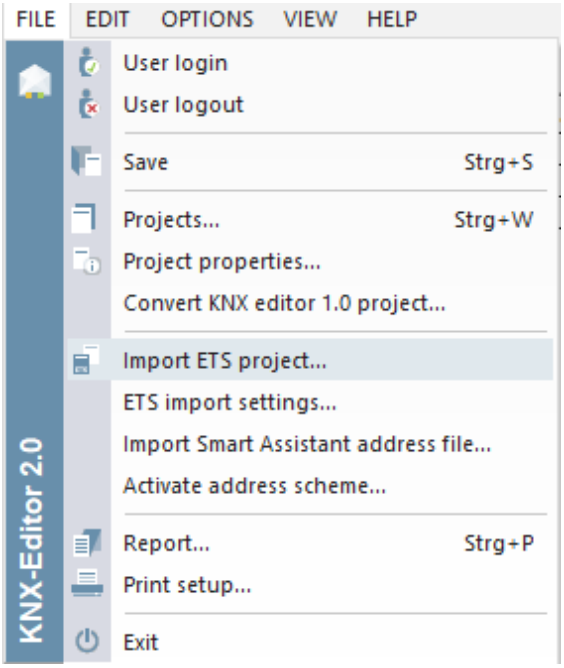
2. Click in the toolbar the button ***Export***.

3. Enter the project name *"Projectname.knxproj"* and click the button ***Save***. The default extension of the file is ***.knxproj***.

4. A dialog indicating the progress of the data transfer opens. After the transfer is completed, confirm the pop up message box ***Operation successful*** by clicking the button ***OK***.

5. The exported ETS project is now available in the selected directory and can be imported in the KNX editor.

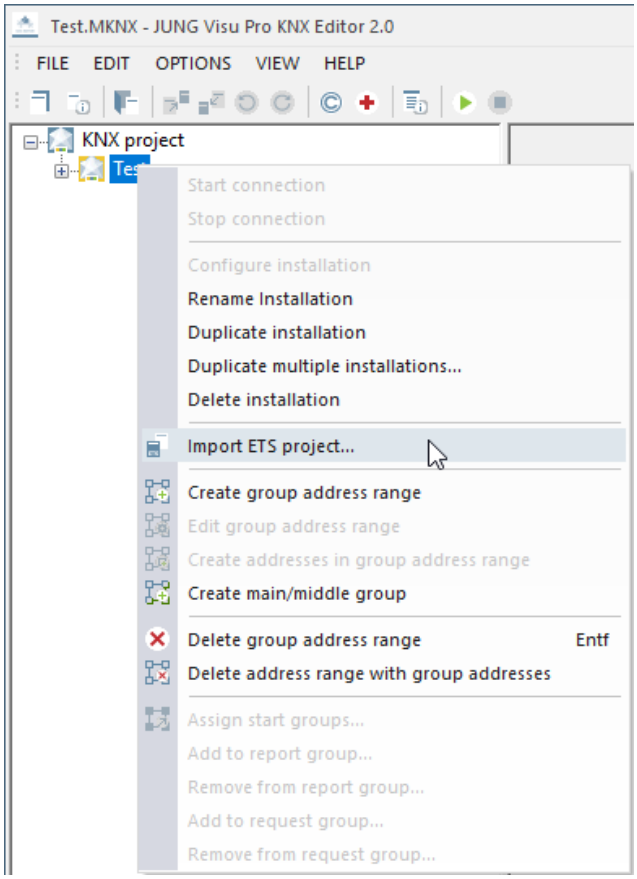
4.12.3 Import of ETS Export Files into the KNX Editor

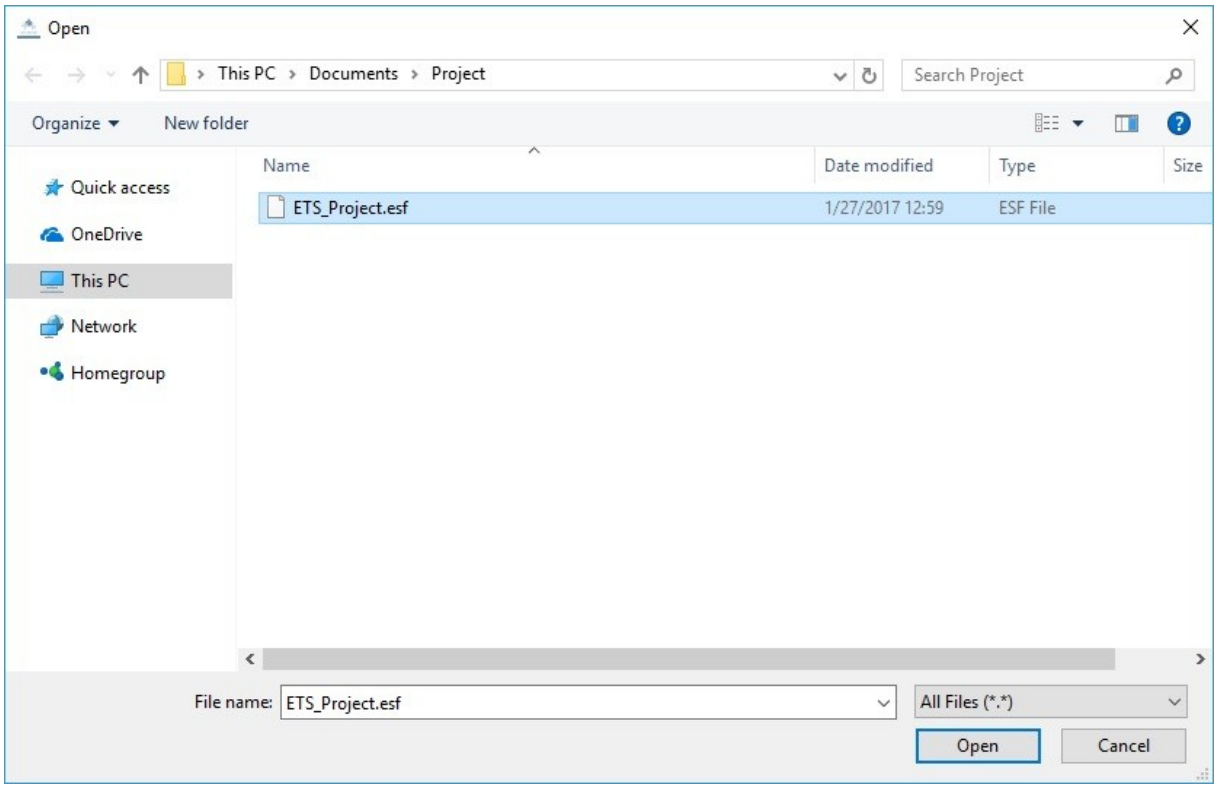


Import ETS project:

First select an installation by clicking on it. Then choose the menu item ***File*** and click ***Import ETS project*** to open the dialog for the selection of the directory.

Alternatively, you can right-click on the installation and select ***Import ETS project*** from the context menu.

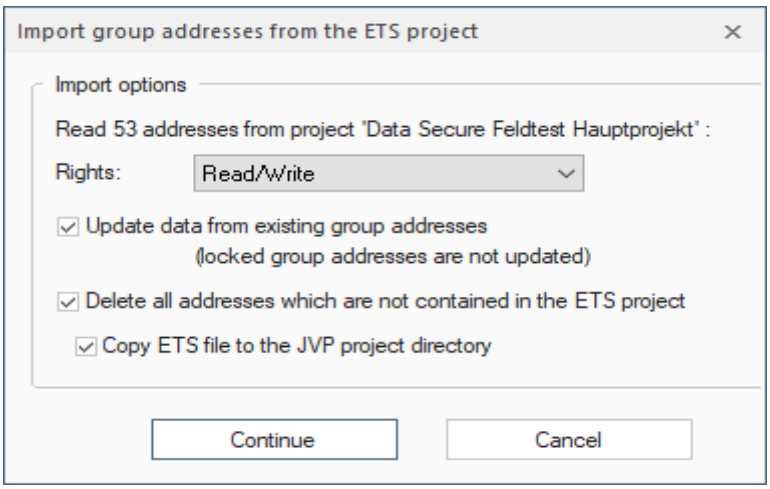




The redirect print reports usually have the extension ***.txt***, the OPC Export-generated files have the extension ***.esf*** and the ETS4 export files have the extension ***.knxproj***.

Select the file to be imported and click the button ***Open***. A dialog for the import opens. Select the project out of the list from the ETS file and confirm with ***OK***.

A dialog for the import of the group addresses opens.



The dialog shows the number of group addresses in the import file. You can set here the access rights which will be assigned to the imported group addresses by default. Since the rights for read and write access cannot be imported, make sure to check them and, if necessary, correct them after the export. But at this step you may select the default rights which are most convenient for the project.

Press ***Continue*** to load the ETS project into the editor.

Import options:

With the check boxes at the import dialog, the following options can be set:

Update data from existing group addresses:

If this option is set, the data of already existing group addresses in the KNX editor will be actualized.

Note: *Enable this option if you want to transfer the changes carried out in the ETS project to the group addresses already contained in the KNX editor (i.e. change of address names).*

Lock and unlock group addresses:

You may want to prevent a data point from being updated. To **lock a group address** for the update, select the checkbox in the column with the padlock icon.

No.	Address	Designation	Value	🔒	Data point type	Unit	Ri
1	00/0/006	Group 6		☑	Boolean	1.001 DPT_Switch	Read/W
2	00/0/007	Group 7		☐	Boolean	1.001 DPT_Switch	Read

To lock several group addresses, click a group address in the column with the padlock icon, hold down the Ctrl key and extend the selection on further group addresses. Then right-click and click in the context menu the option ***Lock address***.

To lock a range of group addresses, click the first group address in the column with the padlock icon, hold down the shift key and click the last group address to be locked. Alternatively, click with the left mouse button the first address to be locked and move the mouse cursor to the last address to be locked. Then right-click and click in the context menu the option ***Lock address***.

To unlock an address for the update, clear the check box or right-click and click in the context menu the option ***Unlock address***.

> ***Note:*** *If you check the option* ***Delete all addresses which are not contained in the ETS project****, the locked group addresses will also be deleted if they are not contained in the ETS project.*

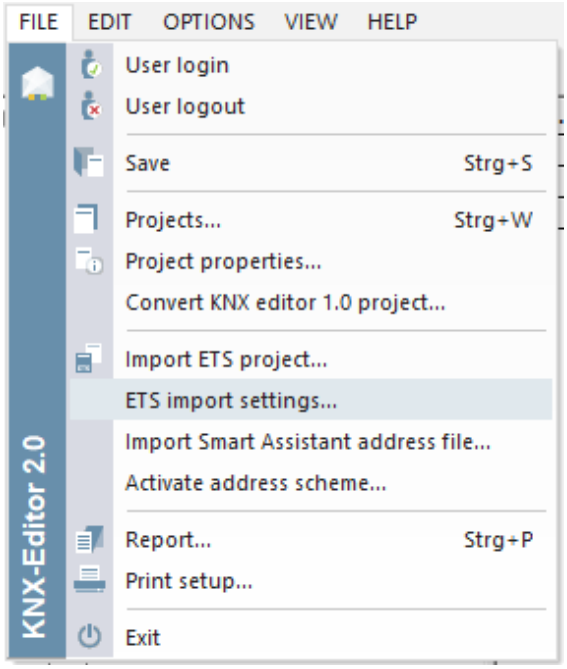
Delete all addresses, which are not contained in the ETS project:

If this option is enabled, all addresses will be deleted which are available in the KNX editor but not in the ETS project.

Copy ETS file to the JVP project directory:

Activating this option places a copy of the imported project file in the JVP project directory.

4.12.4 ETS Import Settings...

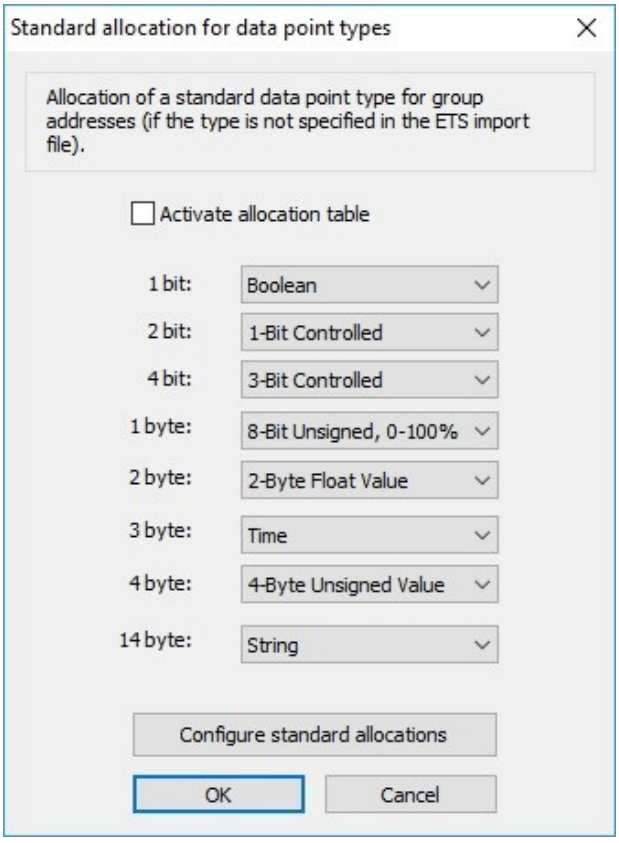


ETS import settings:

Choose the menu item ***File*** and click ***ETS import settings...*** to open the dialog for the standard type declaration for data points.

In this dialog it is possible to activate the declaration table. The decalration table sets standard types for every data point, which has no type specified after the importation of an ETS file.

You can specify a standard data point type foreach available data point length.



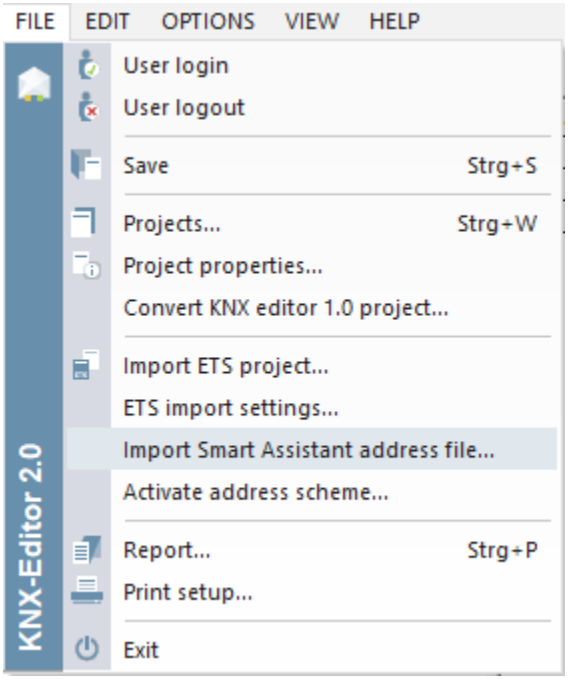
Following data point types are the default values of the declaration table:

1 Bit	Boolean
2 Bit	1-Bit Controlled
4 Bit	3-Bit Controlled
1 Byte	8-Bit Unsigned, 0-100%
2 Byte	2-Byte Float Value
3 Byte	Time
4 Byte	4-Byte Unsigned Value
14 Byte	String

These standard declarations can be restored by clicking the ***Reset to standard declarations*** button.

4.13 Import of Smart Assistant Address Files

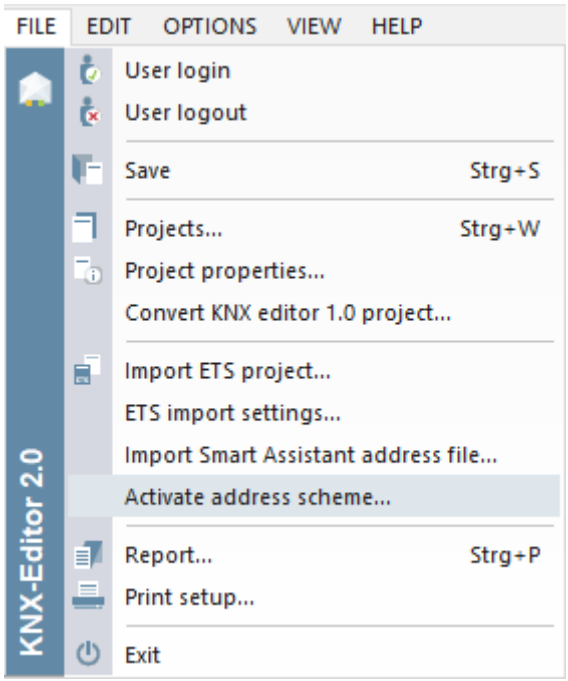
It is possible to import address files, generated by the JUNG Visu Pro Smart Assistant, into the KNX editor.



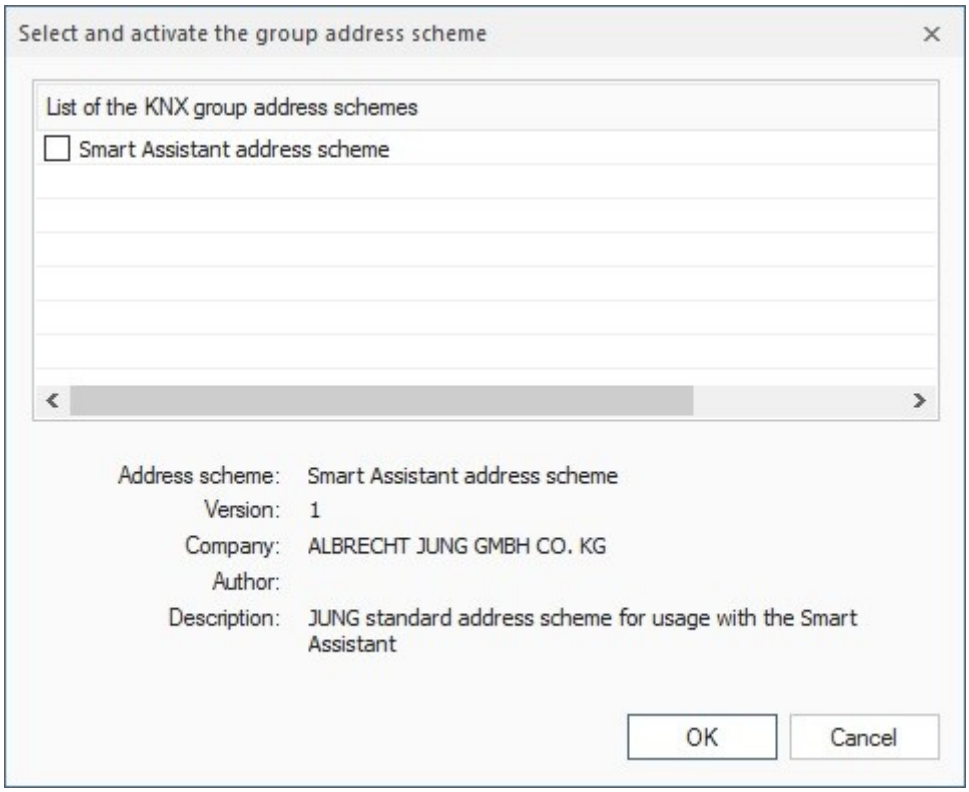
Import of a Smart Assistant address file:

Choose the menu item ***File*** and click ***Import Smart Assistant address file...*** to open the dialog for the selection of the address file.

4.14 Activate Address Scheme

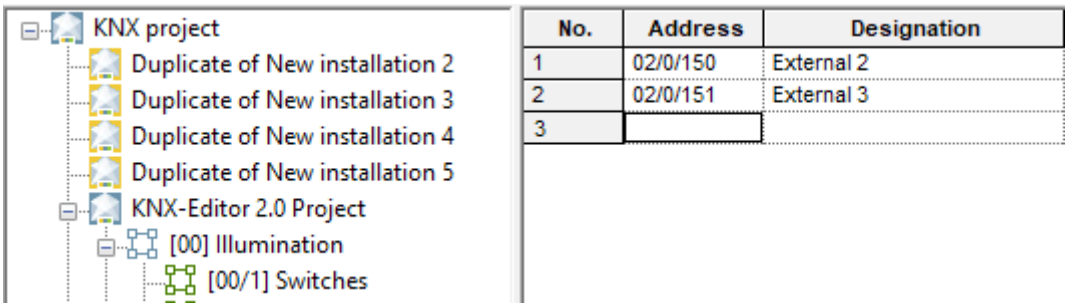


Choose the menu item ***File*** and click ***Activate address scheme...*** to open the dialog for the selection of the address scheme.



This dialog offers schemes, which can be activated to define specific labels for main and middle groups or to set a standard datapoint type for datapoints in specific middle groups. At the moment the only available scheme is the JUNG Visu Pro Smart Assistant scheme.

The JUNG Visu Pro Smart Assistant scheme offers the same structure for your KNX project, which is used when creating a Smart Assistant project. There is no need to define the labels on your own.



4.15 KNX Gateway Settings

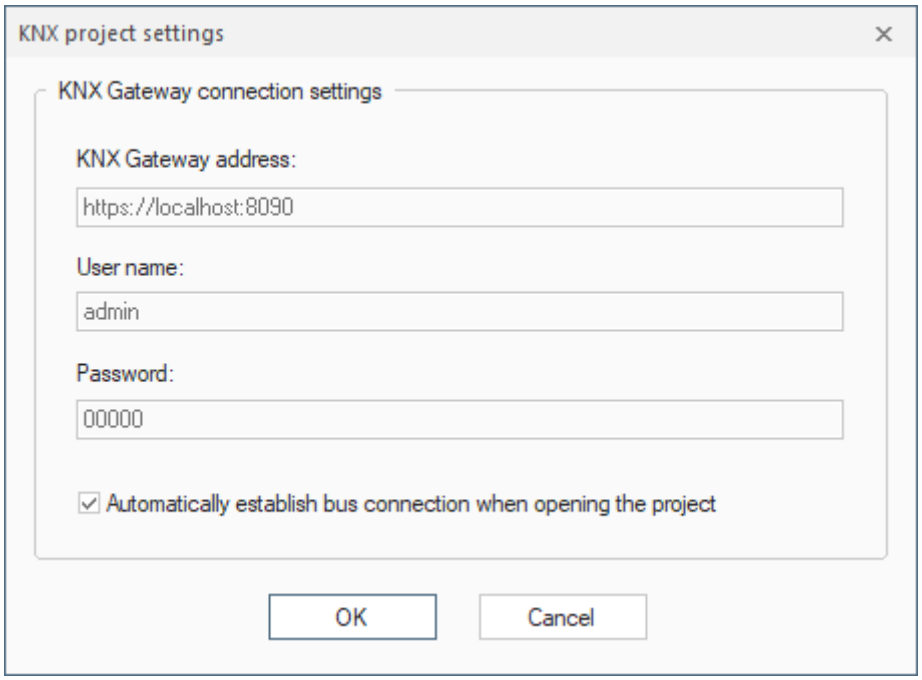
Connect PC to KNX:

To communicate over KNX the JUNG KNX Gateway has to be installed, which uses the Falcon driver from the KNX Association.

Set KNX parameters:

The KNX editor connects to the KNX Gateway, which provides interfaces to connect to the KNX bus.

Choose the menu item ***Options*** and click ***KNX Gateway settings***. The dialog ***KNX project settings*** opens.



Here you can specify the address of the JUNG KNX Gateway, as well as a valid user name and password. Users can be created in the dashboard of your JUNG KNX Gateway.

Start the bus connection once, as described in **4.14 Connect to KNX Bus.** After the connection was successful, you can close it again. Now all KNX interfaces should be available to configure your installations.

Automatically establish bus connection when opening the project:

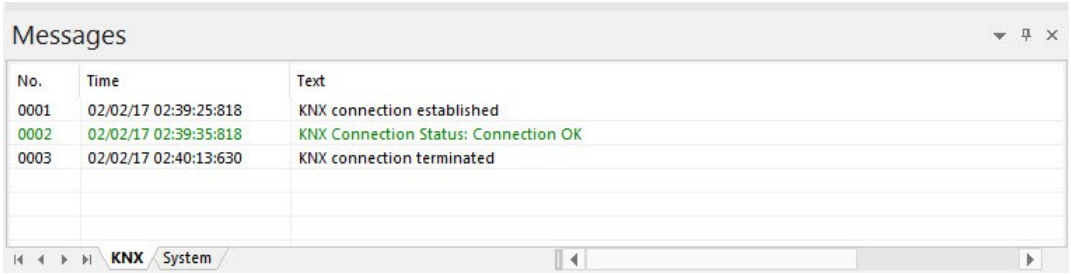
Enable this check box, if you want to start the project automatically, when it is opened. This is useful, if you restart your system and want to open the visualization without having to start the KNX project manually.

4.16 Connect to KNX Bus

To start the connection to the KNX bus, choose the menu item ***Options*** and click ***Start KNX***. Alternatively, click on the symbol in the toolbar.

To interrupt the connection to the bus, choose the menu item ***Options*** and click ***Stop KNX*** or click the symbol in the toolbar.

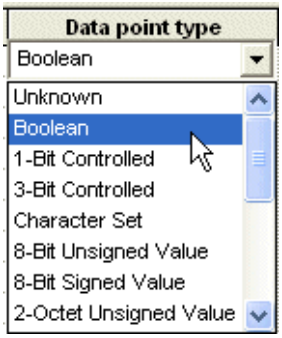
Note: To connect to the bus, you have to configure the installation. This process is described in chapters 4.3.1 and 4.3.2.

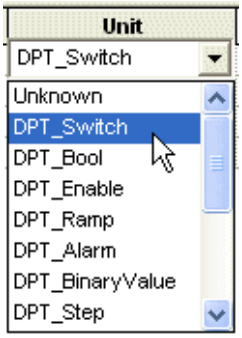


You can see the results of these operations and possibly already received telegrams in the message list.

4.17 Send Telegrams Manually

In the KNX group addresses grid make sure that the columns ***Data point type***, ***Unit***, and ***Rights*** contain the proper value for each KNX group address.



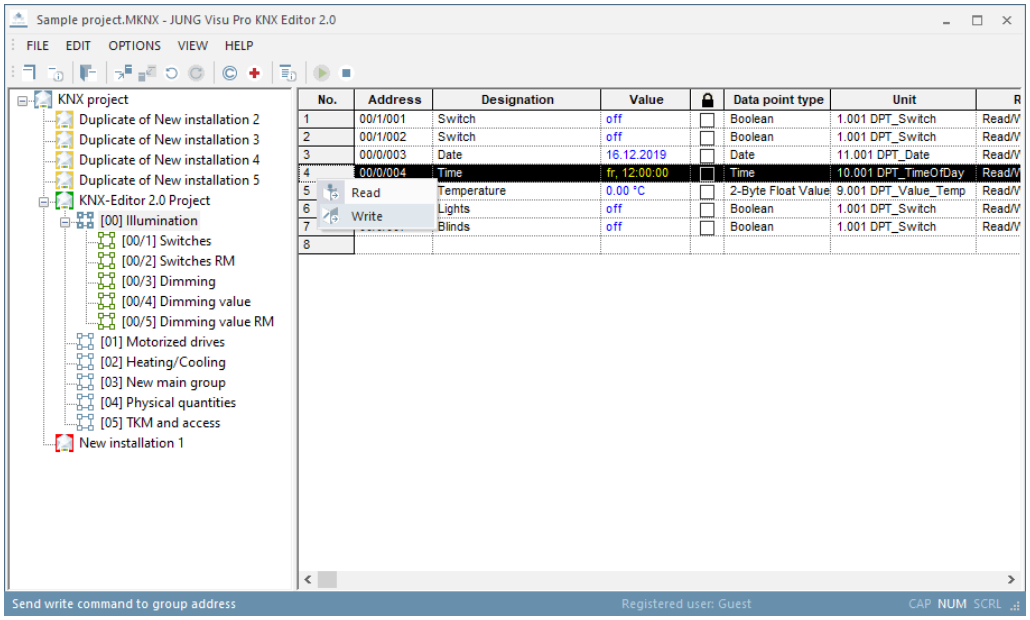


If the KNX group address is properly defined and the write access is allowed, a value can be entered in the column ***Value***. Alternatively, right-click in the value column and select the desired value in the context menu (picture below).

No.	Address	Designation	Value		Data point type	Unit	Rights
1	00/0/001	Date	1/13/2017	☐	Date	11.001 DPT_Date	Read/Write
2	00/0/002	Time	fri, 12:00:00	☐	Time	10.001 DPT_TimeO	Read/Write
3	00/0/003	Temperature	0.00 °C	☐	2-Byte Float Valu	9.001 DPT_Value_	Read/Write
4	00/0/004	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write
5	00/0/005	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write
6	00/0/006	Lights	off	☐	Boolean	1.001 DPT_Switch	Read/Write
7	00/0/007	Blinds	off	☐	Boolean	1.001 DPT_Switch	Read/Write
8							

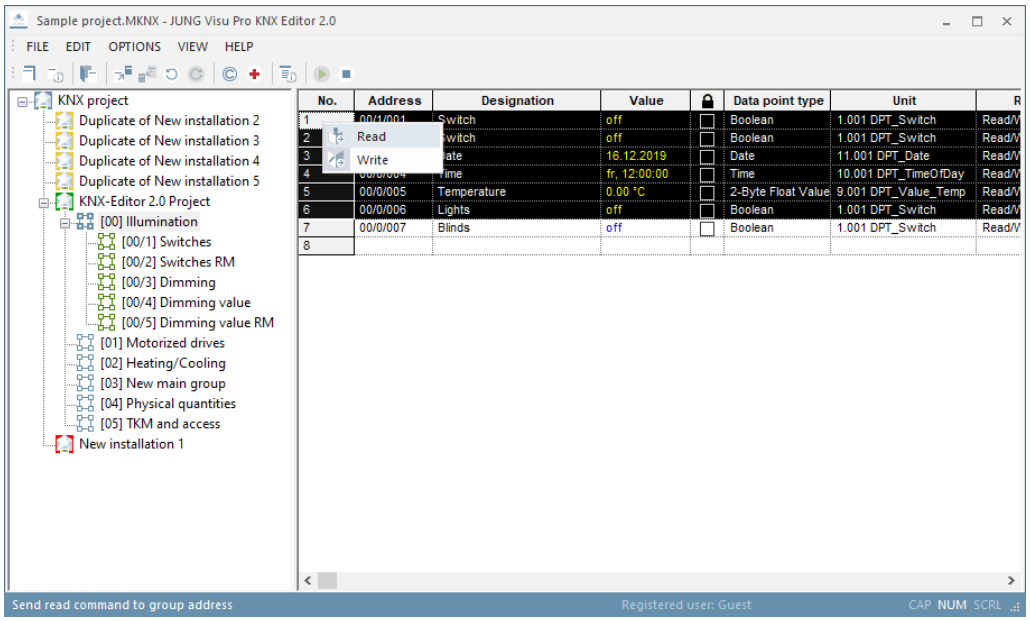
For binary values (type Boolean) use ***0*** for ***off*** and ***1*** for ***on***. For other types just enter the value. The KNX editor will perform the required data conversions.
You can enter an analog value, for example **25.65** for the data point type 2 Octet Float. When the value is to be sent, the content of this column is read, converted into a KNX telegram on the basis of the specified data point type and sent via the bus.

To send the value via the bus, select the entire row of the group address with a left click on the ***No.*** column in the row of the group address. Then right-click and select the desired command, ***Read*** for sending or ***Write*** for querying the value.



After the command has been performed, the telegrams are shown in the message list.

To send several commands at the same time, click the ***No.*** column. Then perform a right-click and select the desired command in the context menu.

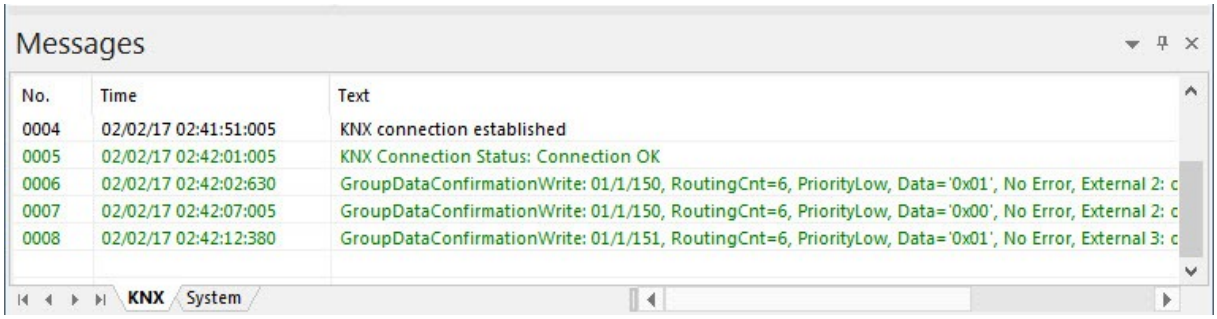


4.18 Message List

The message list shows all KNX telegrams sent and received. When sufficient information regarding KNX data point type and unit has been supplied, the telegrams will be displayed in plain text.

To toggle the visibility of the message list, choose the menu item ***View*** and click ***Messages***. Alternatively, click the symbol in the toolbar.

Choose the menu item ***View*** and click ***Show last message*** to specify that the message list scrolls automatically to the last message.



Meaning of the colors in the message list:

Green: Sent and acknowledged telegrams.

Blue: Received telegrams.

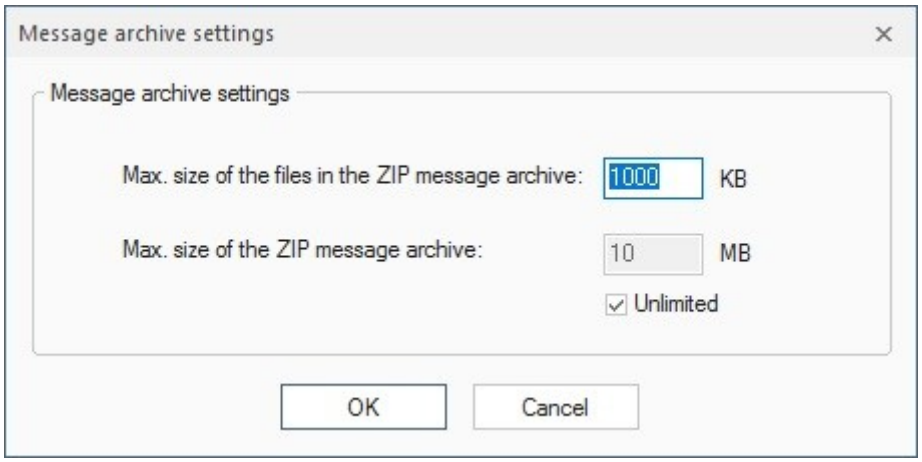
Red: Status or error messages.

Grey: Read requests.

4.19 Logging KNX Messages into Archives

Messages displayed in the message list can be automatically saved in a text file. Choose the menu item ***Options*** and click ***Save KNX telegrams*** to activate or deactivate the automatic logging.

To configure the settings for the message archive, choose the menu item ***Options*** and click ***KNX archive settings***. The dialog ***Message archive settings*** opens.



When the telegram log file reaches a size of about 1 Megabyte, it will be closed and saved into the zip archive **EibMessages.zip** at the directory of the KNX project**:**

(...\stations\local\projects\<projectname>.EIB\EibMessages.zip).

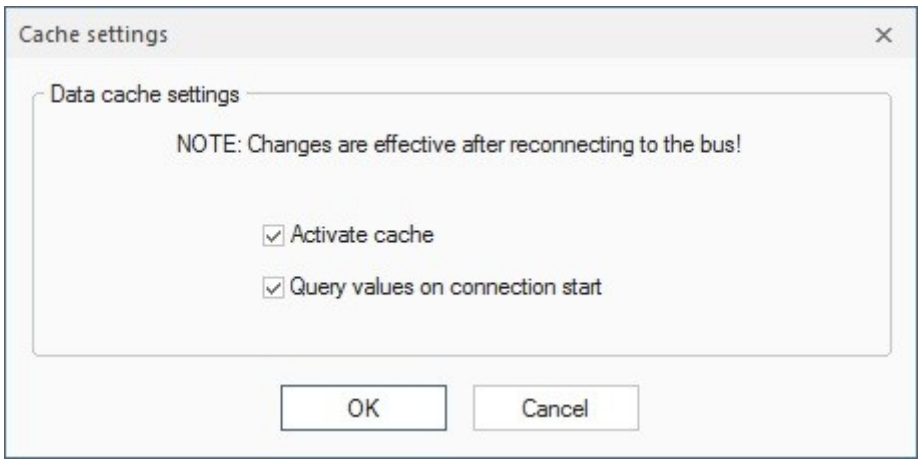
A new log file will be created. The log file will also be closed and saved in the zip archive when the KNX bus has been disconnected.

4.20 Data Cache Settings

The KNX editor comprises a data cache to optimize access to current values of KNX group addresses. Optionally, the cache can be initialized once by querying the bus, but if the values of KNX group addresses are available in the cache, it will update the cache from listening to the KNX without additional queries.

> ***Note:*** *The cache will always be cleared, when the bus is disconnected.*

Choose the menu item ***Options*** and click ***Cache settings***. The dialog Cache settings opens.



Check boxes within the dialog:

Activate cache: Select this check box to activate the cache.

Query values on connection start: When this check box is selected, the cache will query values for KNX group addresses whenever a connection to the KNX bus is established.

> ***Attention:*** *Querying values may produce unexpected results when the access rights for the KNX group addresses have not been properly configured.*
>
> ***Therefore check the settings of the access rights before enabling this option.***

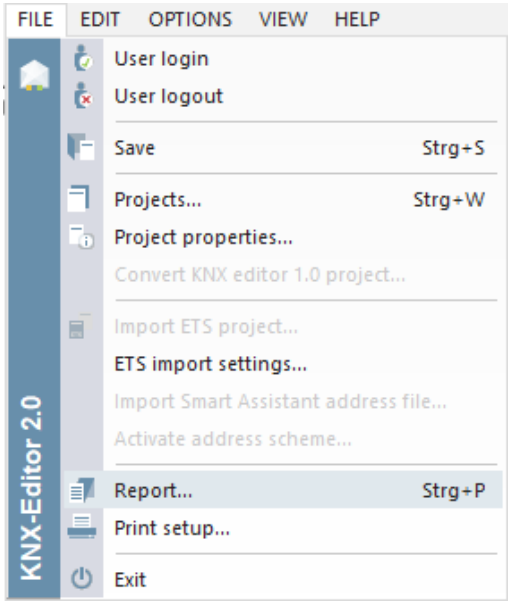
Choose the menu item ***Options*** and click ***Clear cache*** to clear the cache manually.

5 Reports

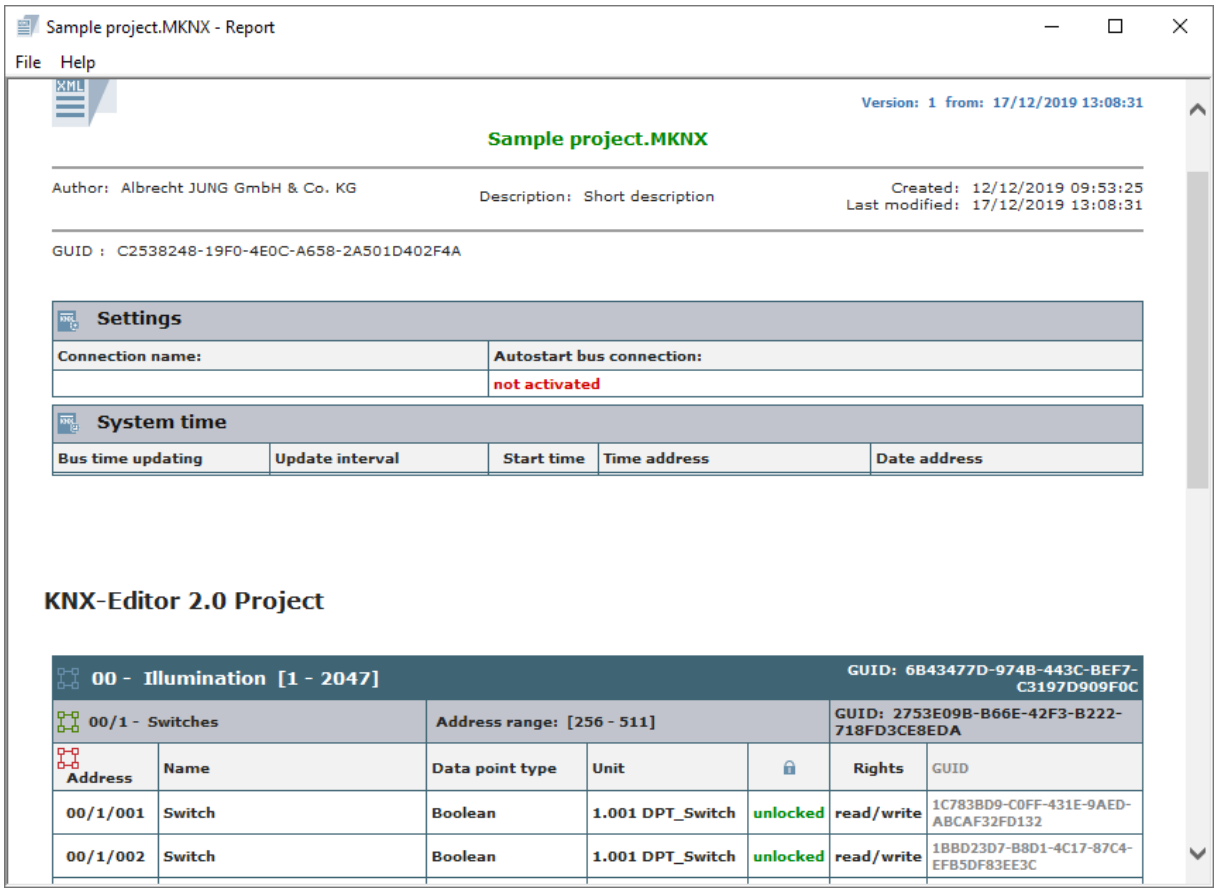
- A report can be created which gives information about the process variables contained in the project and their properties.
- Reports can be printed out (with print preview).

5.1 Create Report

To create a report for the current project, choose the menu item ***File*** and click ***Report***. The dialog ***Create report*** opens. Select the desired options for the report generation and click the button ***Create***.



The data points in the project may be sorted by either names or group addresses.



5.2 Print Preview

The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these , markers the user has visual control over the margin settings.

JUNG Visu Pro
SENEC Editor

Content:

1 TECHNICAL REQUIREMENTS.....2

1.1 Hardware.....2
1.2 Operation Systems.....2

2 INTRODUCTION.....3

2.1 Create a new SENEC Project.....4
2.2 Project Commissioning.....6
2.3 Stop Project.....8
2.4 Change Project Settings.....9
2.5 Rename Project.....9
2.6 Delete Project.....10
2.7 Update Project List.....10

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz
Main memory	1024 MB
Free disk space (additional disk space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

2 Introduction

Click the program symbol to start the control panel of the **visualization system**. In order to clearly distinguish between the different applications, different colors were used. The icon of the **SENEC editor** contains a blue colored bar .

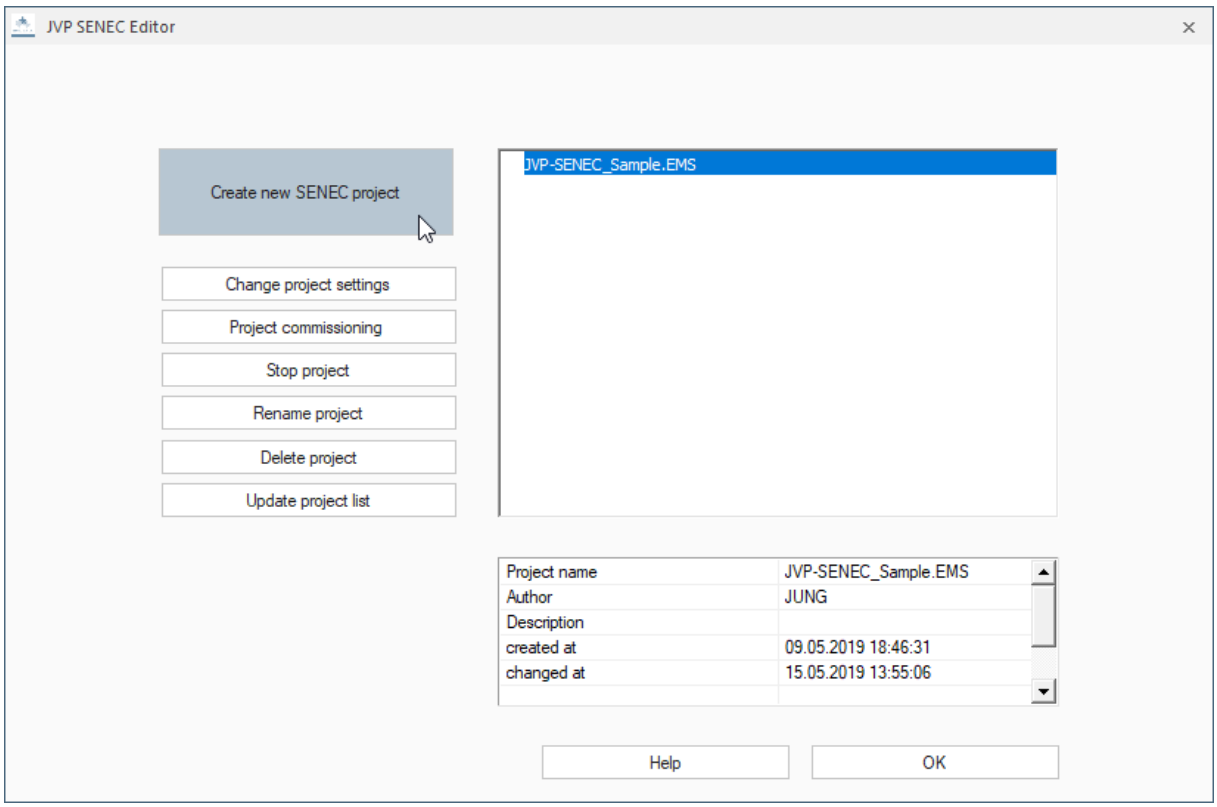
To start the individual function modules, double-click the symbol of the respective modules.

The **SENEC editor** allows you to create Energy Management System (EMS) projects, which enables the data exchange between the JUNG VISU PRO visualization and a SENEC energy storage.

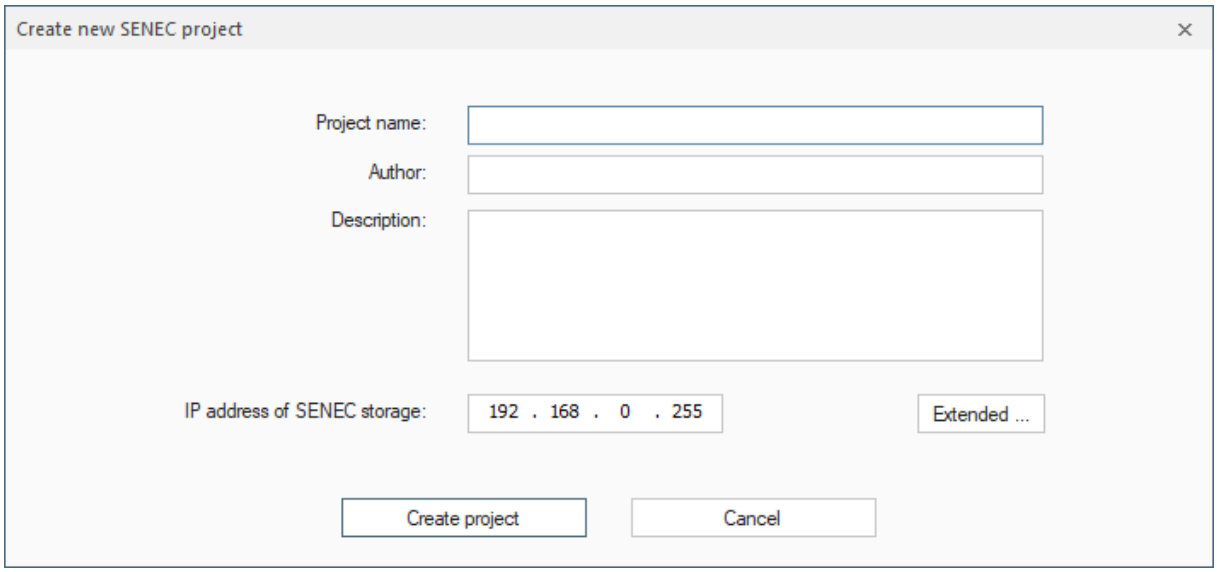
Important note:

The SENEC process connection is not an interface for billing purposes.

Depending on the system, deviations may occur with other visualization systems. Temporary deviations can occur due to transmission intervals and rounding errors – the values of the "**My-Senec**" portal apply.



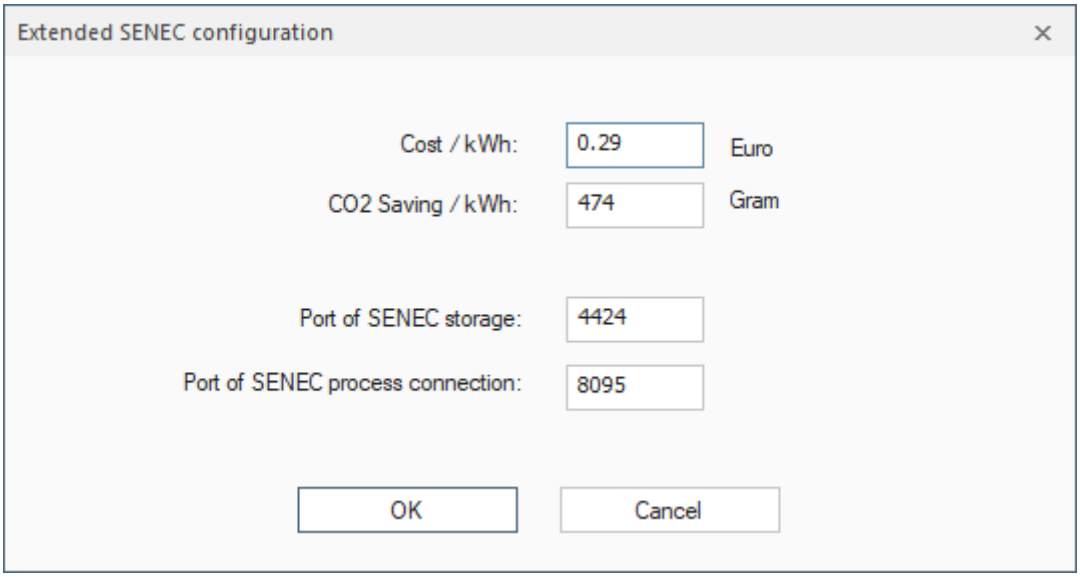
2.1 Create a new SENEC Project



Click the button ***Create new SENEC project***. A dialog opens, enter a project name, an author (optional) and a description (optional).

Enter the corresponding address of your SENEC storage, the default entry is: 192.168.0.255.

A click on ***Extended...*** opens another dialog.



Enter the factors for electricity costs and CO2 savings.

Electricity Tariff

The valid electricity tariff (€ / kWh) is stored here. The factory setting is: 0,29 € / kWh.

Information on the development of electricity prices can be found, for example, in the publications of the Federal Ministry for Economic Affairs and Energy (reference: bmwi.de, 2019).

Factor for Calculating CO2 Savings (g/kWh)

The factory setting is: 474 g/kWh

The value is geared to the published CO2 emission factor of the German electricity mix (extrapolation Federal Environment Agency 2018) (source: umweltbundesamt.de, 2019).

Depending on the system, deviations may occur with other visualization systems! The calculated values do not serve billing purposes!

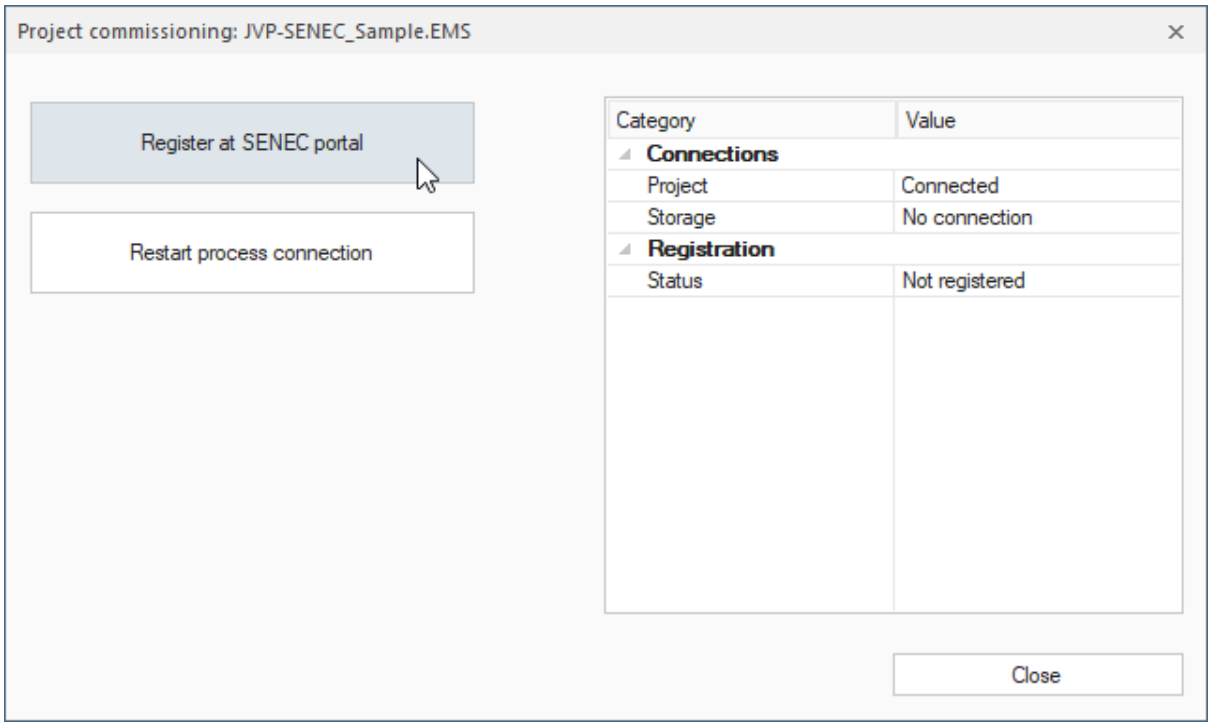
Communication Ports

Factory setting for the SENEC storage: 4424
Factory setting for JVP process connection: 8095

The factory-set port for the SENEC process connection usually does not have to be changed, the registered port of the SENEC storage if required.

When the SENEC process connection is started for the first time or when the port of the JVP process connection is changed, the user has to confirm the access to the set port twice.

2.2 Project Commissioning



The first time the JUNG VISU PRO Visualization is connected to the SENEC storage, JVP registers itself with the battery storage. The data exchange between JVP and storage must now be permitted, i.e. confirmed, by the owner of the storage. The confirmation takes place in the SENEC customer portal. Once the confirmation has taken place, data can be exchanged between JVP and the battery storage.

The status of the registration can be tracked in the SENEC editor and a new release requirement can be sent to the portal if necessary.

Click on ***Project commissioning*** to open a dialog in which you can observe the connection status and the release status.

Connection status: "Connections"

Project:

Status of the connection between JVP visualization and SENEC process connection:

---	No data can be received from the SENEC process connection.
Connected	Data can be received from the SENEC process connection.

Storage:

Status of the connection between the SENEC process connection and the SENEC energy storage:

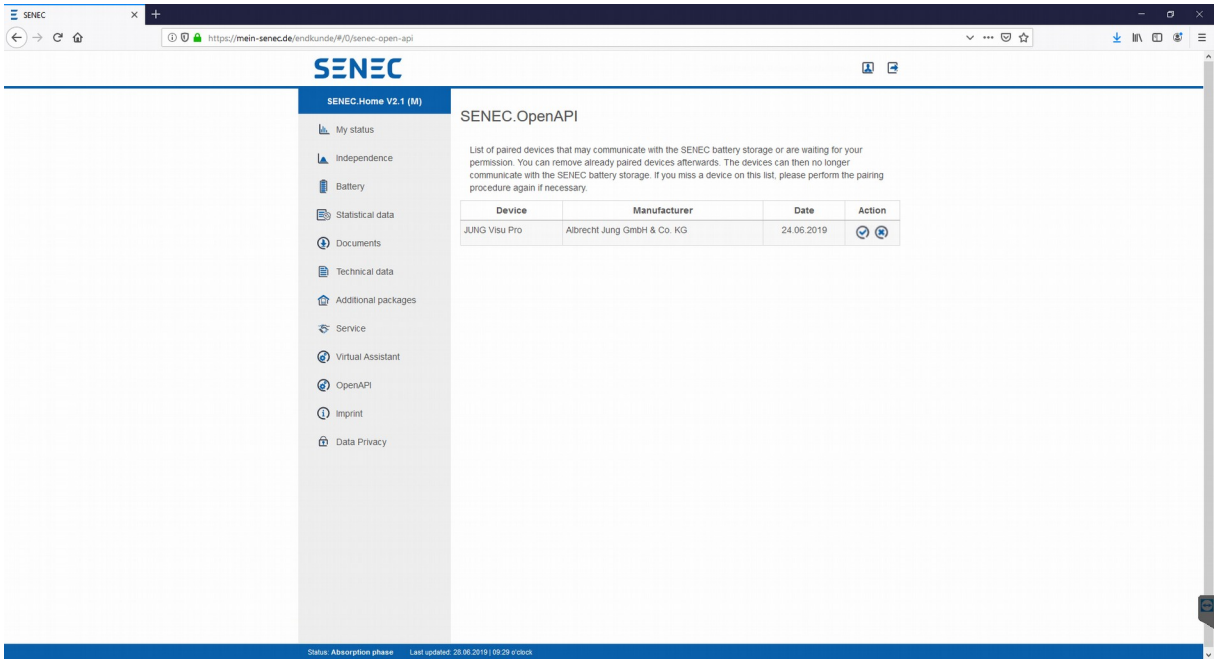
---	The status of the connection is unknown.
Connected	The battery storage is accessible and responds to queries.
No connection	The battery storage is not accessible respectively does not respond to queries.

Release status: "Registration”

Status

Status of the registration process. Possible values are:

Not registered	The JUNG VISU PRO Server has not yet registered with the battery storage.
Wait for confirm	The JUNG VISU PRO Server has registered with the battery storage and is waiting for a confirmation by the user in the SENEC customer portal. Although it is possible to communicate with the storage, the consumption data of the battery storage can only be readout if the release has been effected by the user.



The connection request must be confirmed in the portal:

Registered	The JVP is registered, the release has been effected. Communication with JUNG VISU PRO has been set up successfully – the storage sends its data encrypted to the JUNG VISU PRO server.

Command to the SENEC process connection

Register at SENEC portal

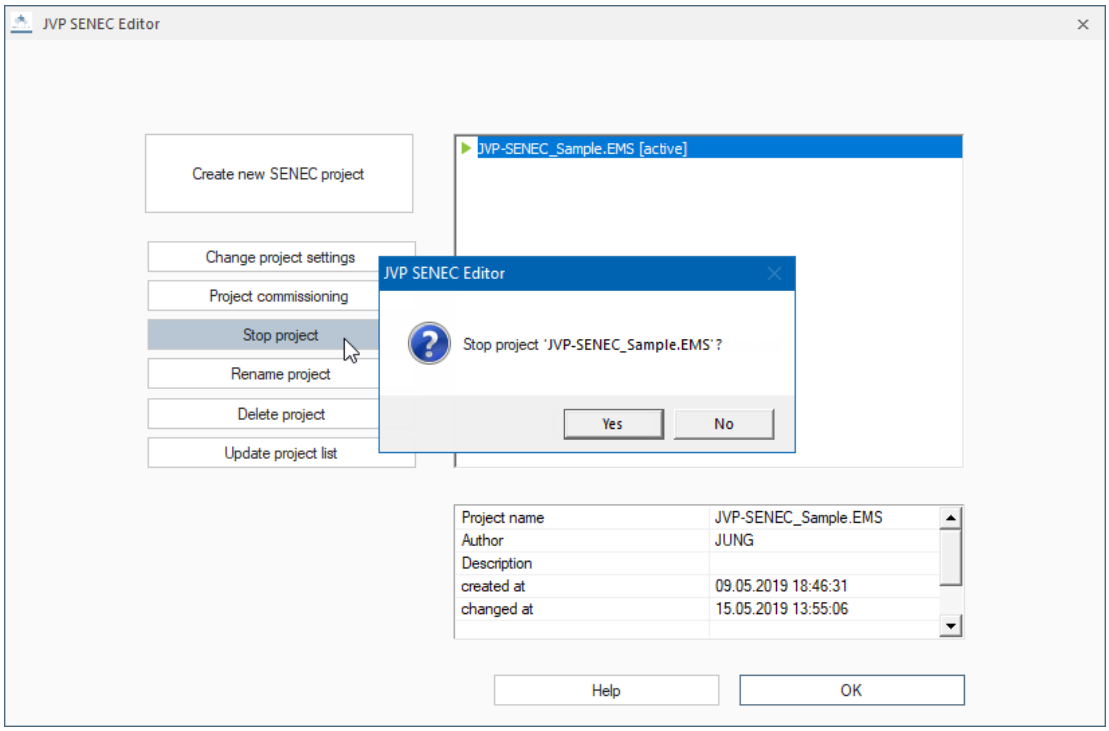
The button ***Register at SENEC portal*** can be used to request a new registration in the SENEC customer portal. JVP then waits for the user's approval in the SENEC customer portal. The data of the storage can only be read again if the release has been effected by the user.

Restart process connection

The button ***Restart process connection*** is used to restart the SENEC process connection.
This may be necessary, for example, if the IP address or the port of the SENEC storage has changed (after updating the data in the ***Change project settings*** dialog).

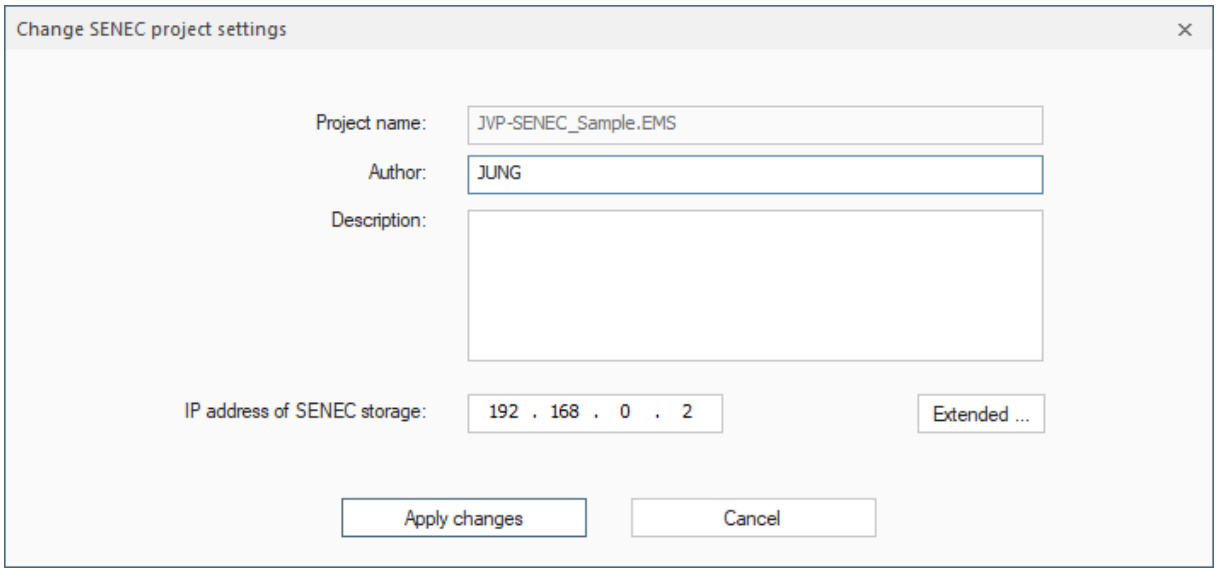
You will recognize a running project by the green arrow:

2.3 Stop Project



Clicking the ***Stop project*** button closes the connection between JVP visualization and SENEC process and stops the project.

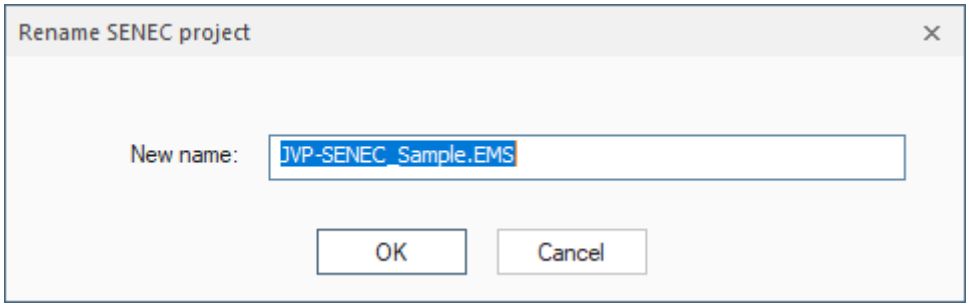
2.4 Change Project Settings



Clicking on ***Change project settings*** opens a dialog in which you can edit your project data. It is not possible to change the name of the project here. A click on ***Apply changes*** saves your entries.

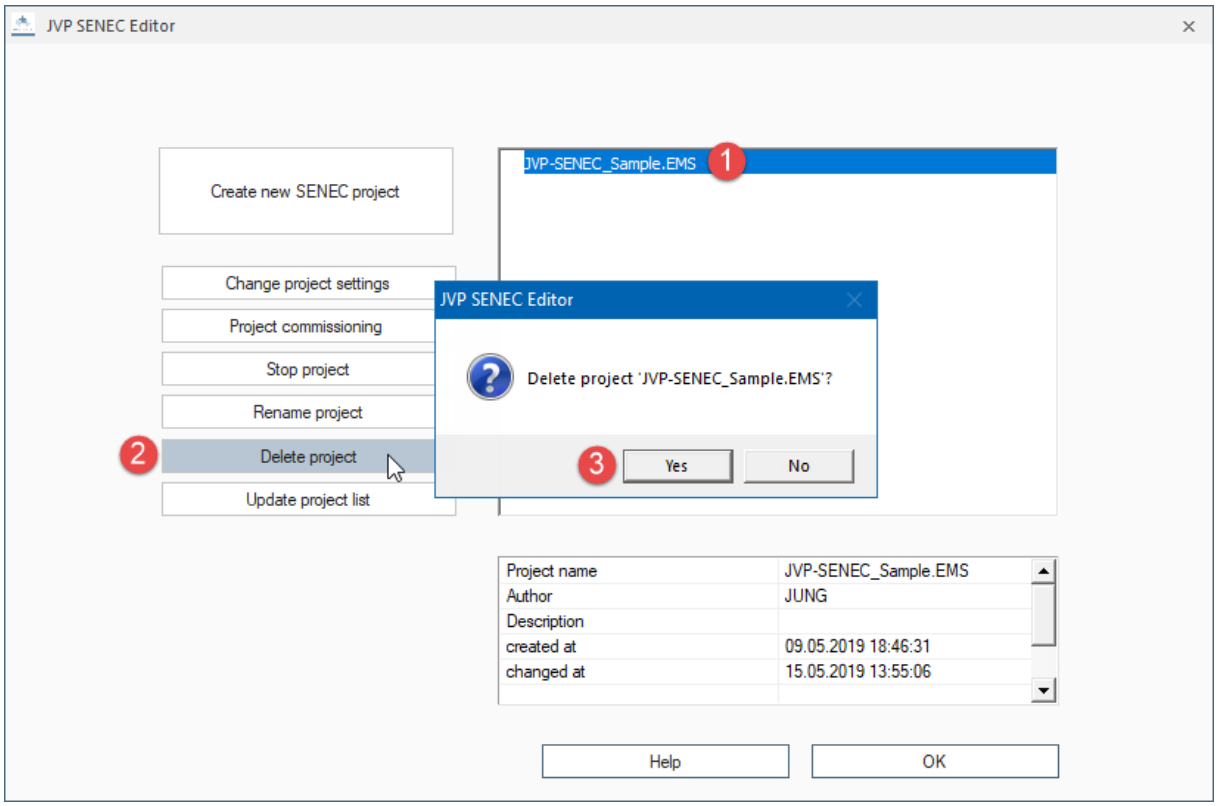
By clicking the ***Extended...*** button you can change the factors for electricity costs and CO2 savings as well as the port settings.

2.5 Rename Project



Clicking the button ***Rename project*** opens a dialog in which you can change the name of the project. A click on ***OK*** saves the changes.

2.6 Delete Project



1. Select a project.
2. Click the button ***Delete project***.
3. Answer the confirmation prompt with the ***Yes*** button to complete the operation.

2.7 Update Project List

By clicking ***Update project list*** project changes made outside the SENEC editor can be updated. For example, if an EMS project has been imported in the control panel and the SENEC editor is still active, the imported project will not be displayed until you click ***Update project list***.

Note:	Depending on the system, the values displayed in the visualization may differ from the values displayed in the SENEC portal.

JUNG Visu Pro
Device Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....8

4 FUNCTIONS....9

4.1 Project Organizer....9
4.2 Process Interface....15
4.2.1 Create Process Interface....15
4.2.2 Start and Stop Process Interface....17
4.2.3 Edit Process Interface....18
4.3 Data Point Folders....19
4.3.1 Create and Delete Data Point Folders....19
4.3.2 Rename Data Point Folder....19
4.3.3 Edit Data Points....20
4.4 Running the "UDP commands" Process Interface....20
4.5 Message List....21

5 REPORTS....23

5.1 Create Report....23
5.2 *Print Preview*....24

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a registered trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 Bit per Pixel.
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

From a users point of view, the device editor provides new process interfaces, particularly for a range of IP-capable devices. It is useful for the integration of multimedia applications in building and home automation systems. The device editor provides more connectivity to other devices with custom protocols.

From a software developers point of view, the device editor also provides an environment that allows to extend the system by new process interfaces, as far as the data protocol can be reasonably described in a script. The device editor provides an application program interface (API) to be used in the script, and a user interface for its data which are described in an XML file. Each process interface with XML and script files is simply read from a file directory which can be copied from one computer to another and run as a plug-in in an own thread.

The document does not cover the development point of view, but only the application of the device editor with the "UDP commands" process interface. The "UDP command" process interface is automatically installed by the setup program. It is assumed that more and more process interfaces will be available as plug-ins to the device editor, either developed by us or contributed by others.

After the installation of the visualization system a program group is created in the Start Menu. You can adopt the default settings or change them manually during installation.

You can start the control panel of the visualization system from the WINDOWS program manager. In order to clearly distinguish between the different applications, different colors were used. The icon of the **device editor** contains a blue colored bar . Projects of the device editor have the extension "**SDV"**.

To start an application, select the list entry ***Function Modules*** in the sidebar of the control panel and double-click the icon of the application you wish to start.

Proceeding:

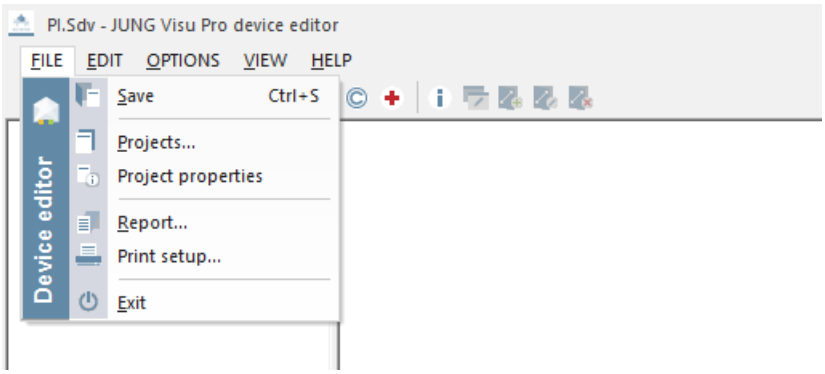
1. Create a device editor project.
2. Create a process interface.
3. Edit the process interface.
4. Start the process interface.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

3 User Interface

3.1 Menu

Menu item *File*

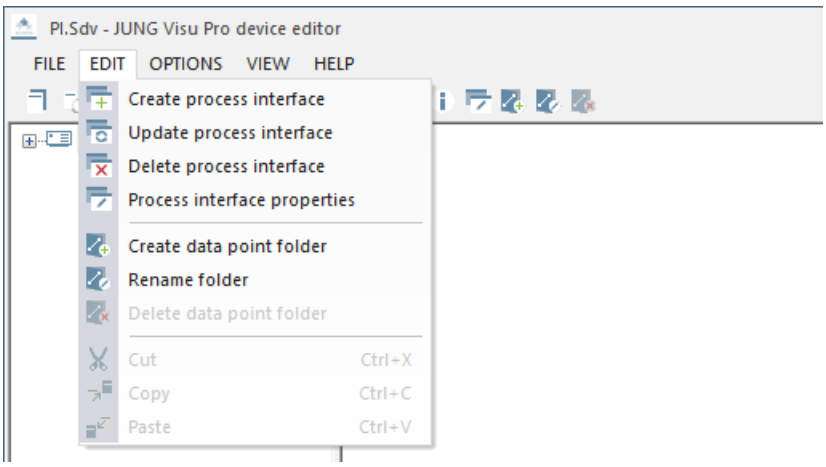


Save: Save current project. Shortcut Strg + S.

Projects: Opens the project organizer dialog. In this dialog projects can be created and maintained.
Project properties: Opens a dialog to define or update general project properties.

Report: Creates a report of the current project.
Print setup: Selection of printer and setting of printer properties.
Exit: Closes the device editor and saves the open project. Shortcut ALT + F4.

Menu item *Edit*



Create process interface: Opens the dialog to create a new process interface based on a process interface type.

Update process interface: The implementation of the process interface (by means of script and description files) may have been changed, with insertion or deletion of data points or resource data. In this case, the update function can be used to update the process interface in the editor accordingly.

Delete process interface: Deletes the selected process interface.

Process interface properties: Opens a dialog to edit the process interface properties, including the option to automatically connect the selected process interface when the project starts.

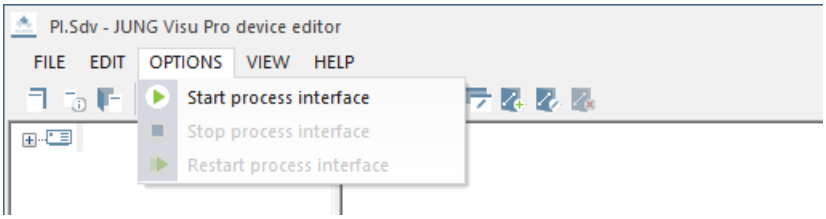
Create data point folder: Opens a dialog to create a new data point folder, with options defined by the current process interface type.

Rename folder: Rename the selected folder, if allowed.

Delete data point folder: Delete selected data point folder.

Cut, Copy, Delete: Standard editing options.

Menu item *Options*

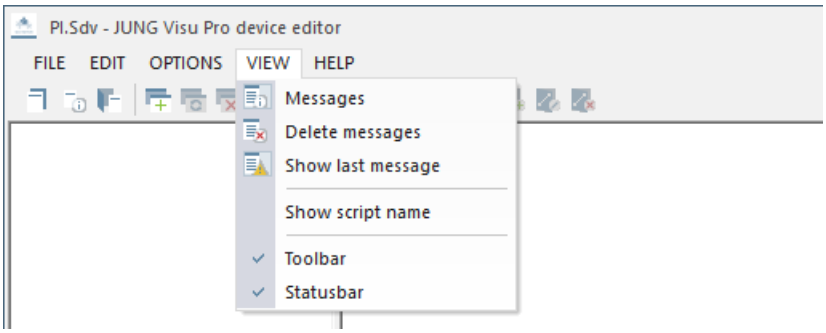


Start Process interface: Starts the script for the selected process interface.

Stop Process interface: Stops the script for the selected process interface.

Restart Process interface: Restarts the script for the selected process interface.

Menu item *View*



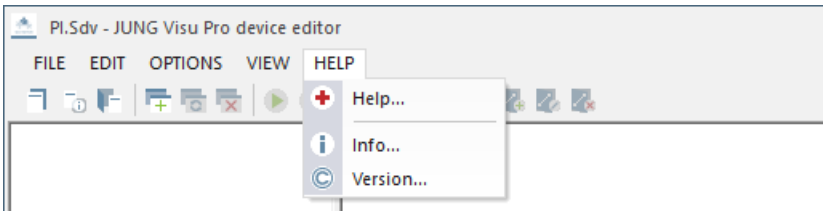
Messages: Shows/hides message lists.
Delete messages: Deletes the contents of the message list.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

Show script name: Besides a display name, folders and folder items have internal names which are used in the implementation script of the process interface to access the respective items in the project. Turn on/off the display of these names.

Toolbar: Shows or hides the toolbar.
Statusbar: Shows or hides the statusbar.

Menu item *Help*



Help: Opens the help file. If a process interface is selected, it opens the help file of the process interface.

Info: Shows product and license information.

Version: Shows version and copyright information.

3.2 Toolbar

Projects: Opens the project organizer dialog. In this dialog projects can be created and managed.

Project properties: Opens a dialog to define or update general project properties.

Save: Saves current project. Shortcut Ctrl + S.

Create process interface: Opens a dialog to create a new process interface based on a process interface type.

Update process interface: The implementation of the process interface (by means of script and description files) may have been changed, with insertion or deletion of data points or resource data. In this case, the update function will update the process interface in the editor accordingly.

Delete process interface: Deletes selected process interface.

Start process interface: Starts the script for the selected process interface.

Stop process interface: Stops the script for the selected process interface.

Version: Shows version and copyright information.

Help: Opens the help file.

Info: Shows the product information.

Process interface properties: Shows the properties of the selected process interface.

Create data point folder: Creates a data point folder in the selected process interface

Rename data point folder: Rename the selected data point folder.

Delete data point folder: Deletes the selected data point folder.

4 Functions

4.1 Project Organizer

The project organizer dialog opens automatically when the device editor is started. To open the project organizer manually in the device editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

In the project organizer you can create new projects and open, close, delete or rename existing projects.

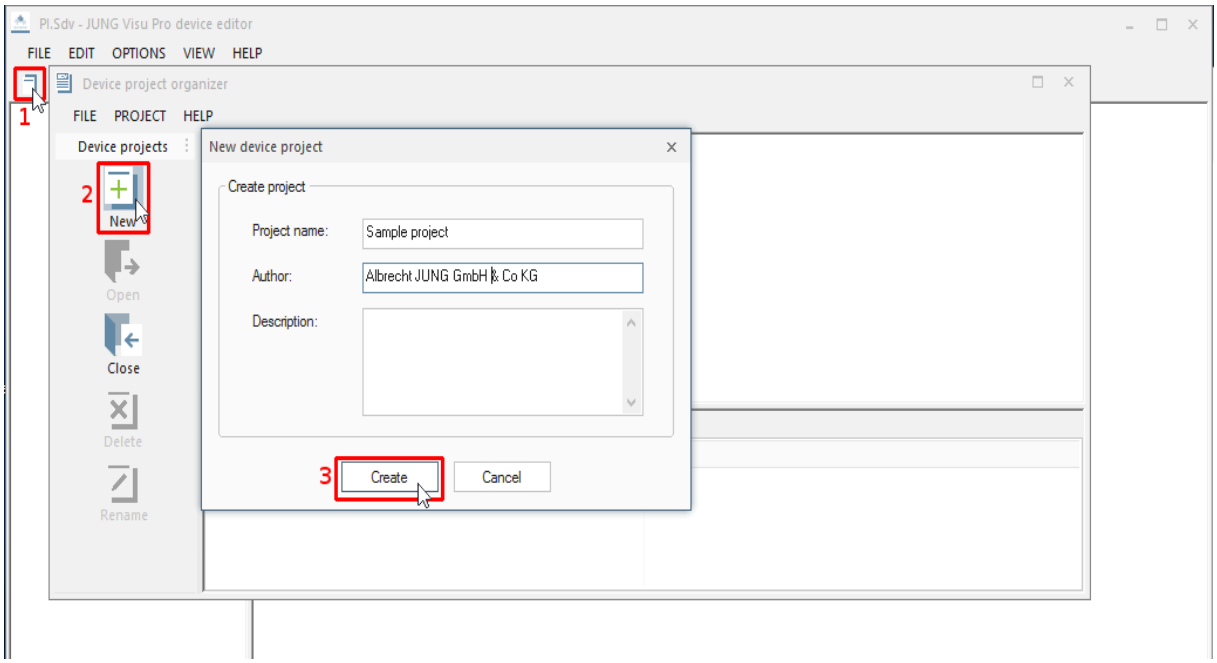
You can use the buttons in the sidebar of the project organizer or the entries under the menu item ***Project***.

The project organizer provides the following options:

- **New**: Create new project.
- **Open**: Open an existing project in the device editor.
- **Close**: Close the current project.
- **Delete**: Delete the selected project.
- **Rename**: Rename the selected project.

Create Project

To create a new project, click the button ***New*** in the sidebar or choose the menu item ***File*** and click ***New***. The dialog ***New device project*** opens.



Enter the following information:

Project name: Enter a unique name for the project.

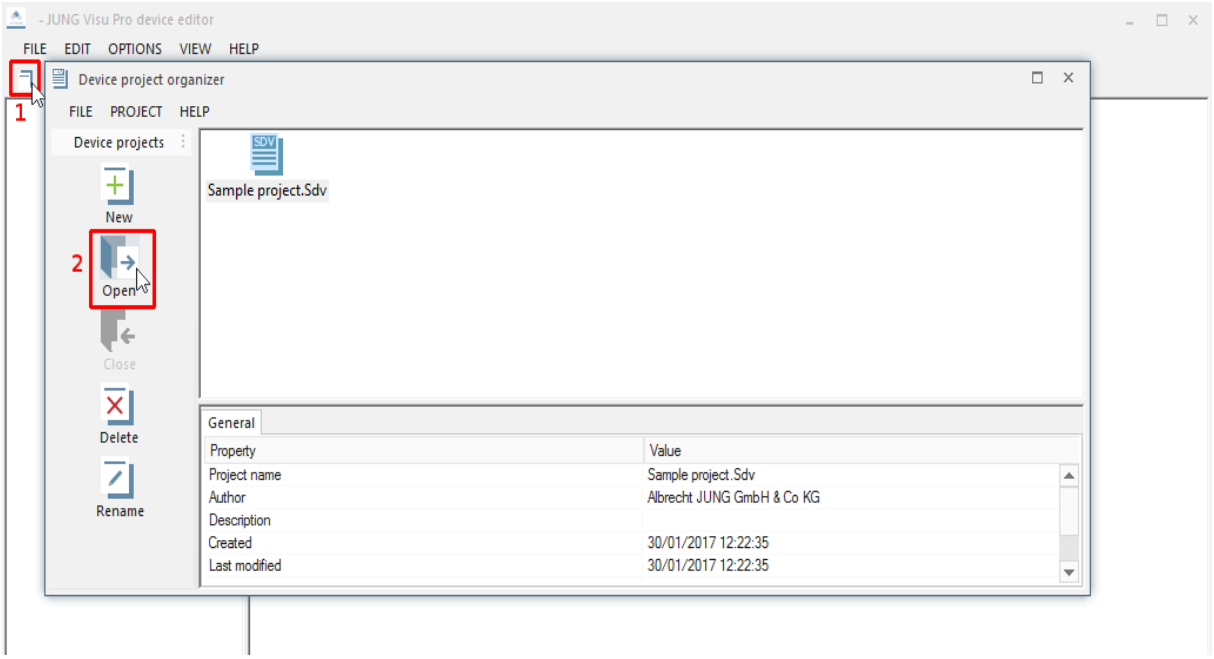
Author and description: These entries are not mandatory and can be changed later.

Create: After entering at least a name for the project, click the button ***Create*** to complete the creation of the device project. The new project is now available in the project list of the project organizer.

Open Project

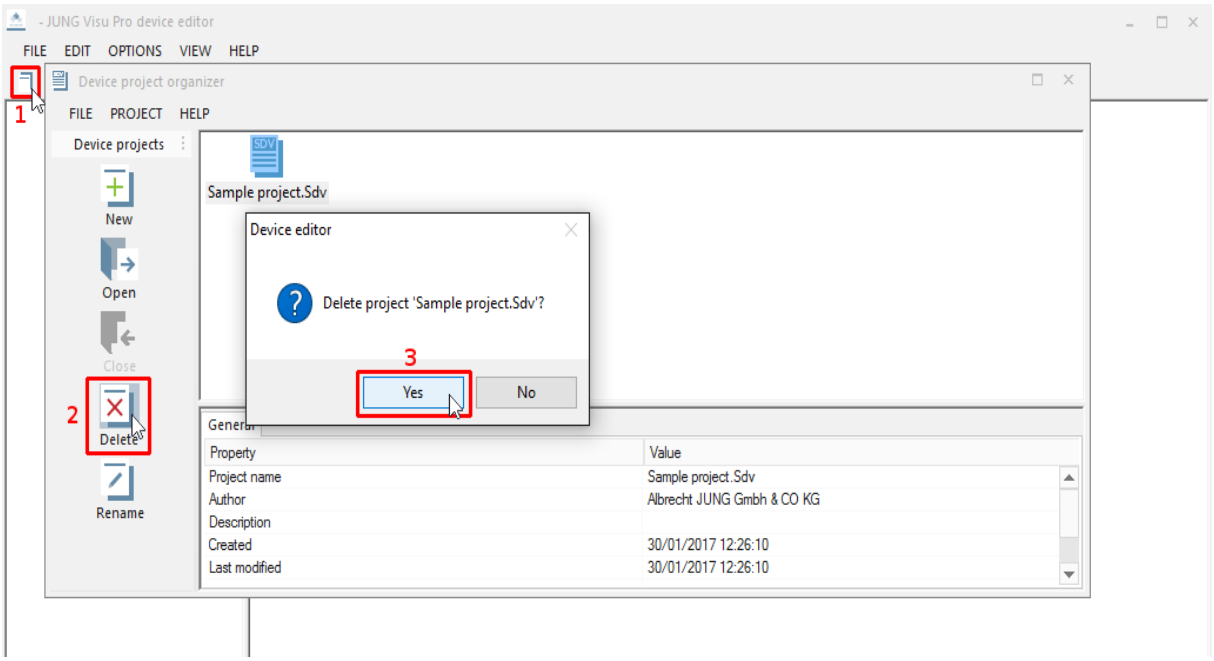
To open the project in the device editor,

- double-click the project symbol in the project list or
- select the project and click the button ***Open*** in the sidebar or
- choose the menu item ***Project*** and click ***Open***.



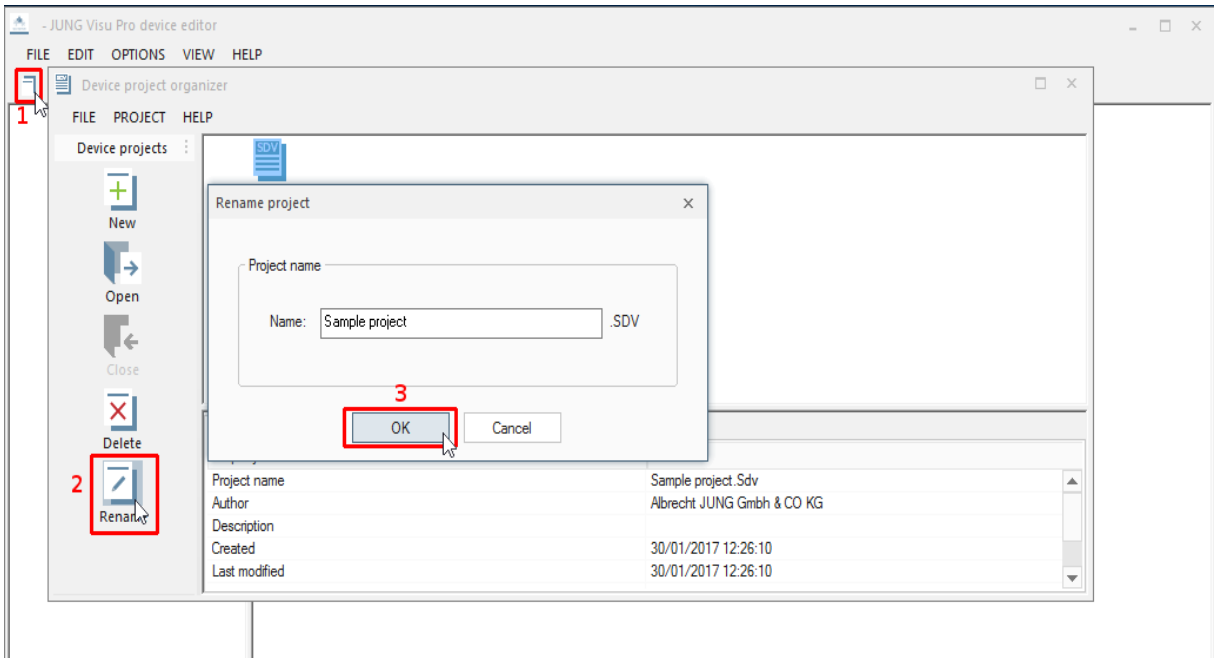
Delete Project

To delete a project, select the project and click the button ***Delete*** in the sidebar or choose the menu item ***Project*** and click ***Delete***. Answer the confirmation prompt with the button ***Yes*** to complete the operation.



Rename Project

To rename a project, select the project and click the button ***Rename*** in the sidebar or choose the menu item ***Project*** and click ***Rename***. The dialog ***Rename project*** opens.

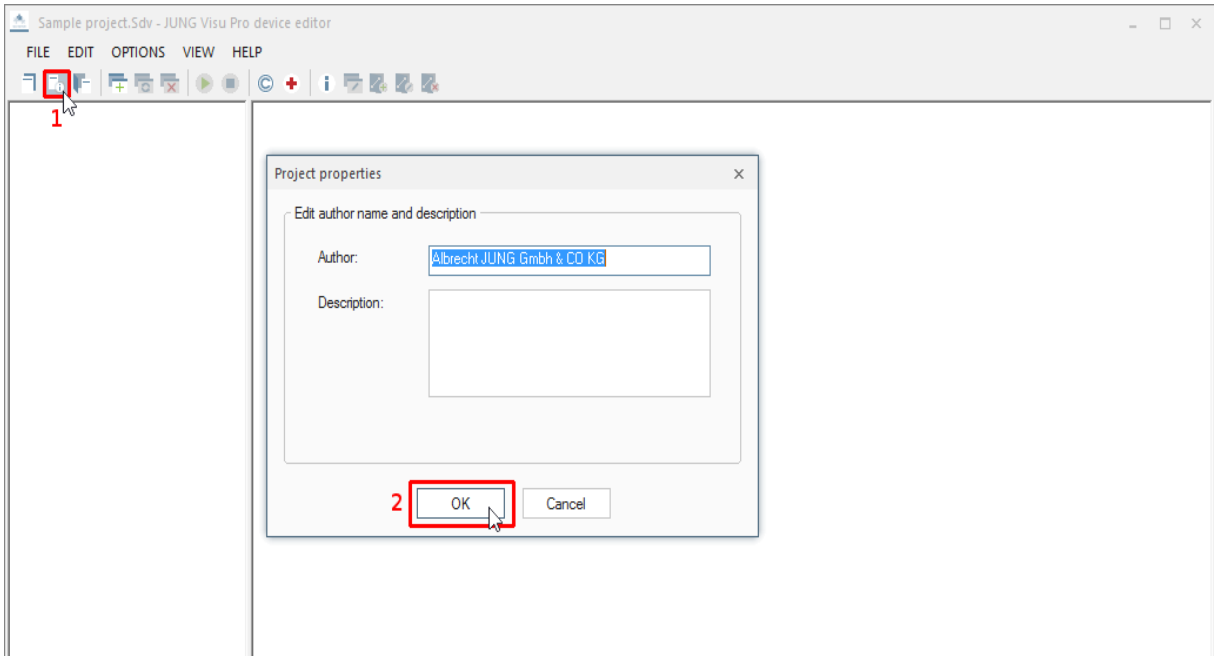


Enter the new name for the project and click the button ***OK*** to close the dialog and submit the change. The extension ***.SDV*** will be added automatically.

> ***Note*** *that a project can only be renamed if it is not currently opened.*

Edit Project Properties

To edit project properties, choose in the device editor the menu item ***File*** and click ***Project properties*** or click the button in the toolbar. The dialog ***Project properties*** opens.

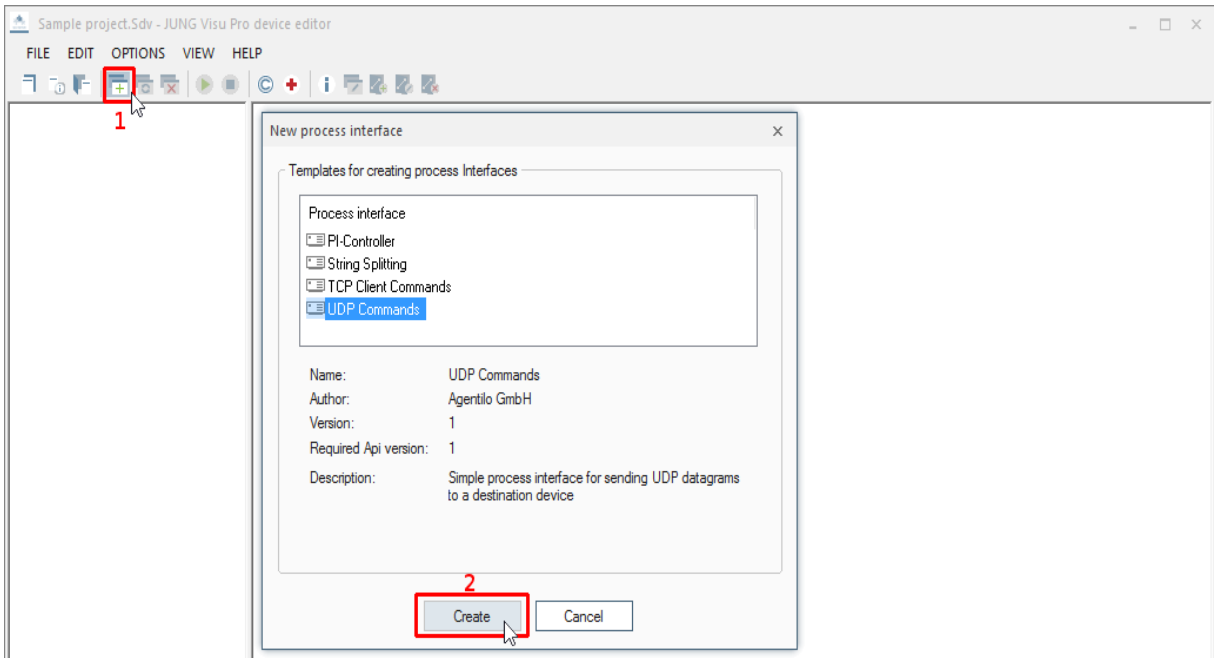


Edit the project properties (author and description) and click the button ***OK*** to commit the entries.

4.2 Process Interface

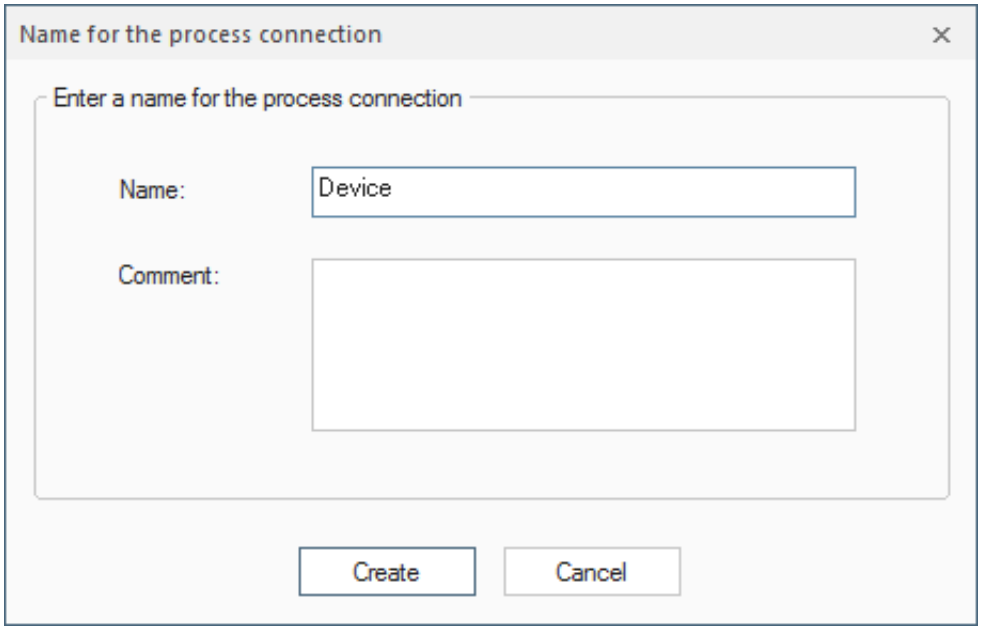
4.2.1 Create Process Interface

To define process interfaces of the devices project, choose the menu item ***Edit*** and click ***Create process interface*** or click the button in the toolbar. The dialog ***New process interface*** opens.



The dialog shows the available process interface types and their properties: name, author, version, a short description and the version of the API (Application Programming Interface) the process interface requires from the device editor as its programming environment. The API version provided by the devices editor is displayed in the info dialog (menu item ***Help*** > ***Info***).

Click the button ***Create*** to create a process interface based on the selected process interface type. A dialog opens where you can specify the name of the process interface and a comment.

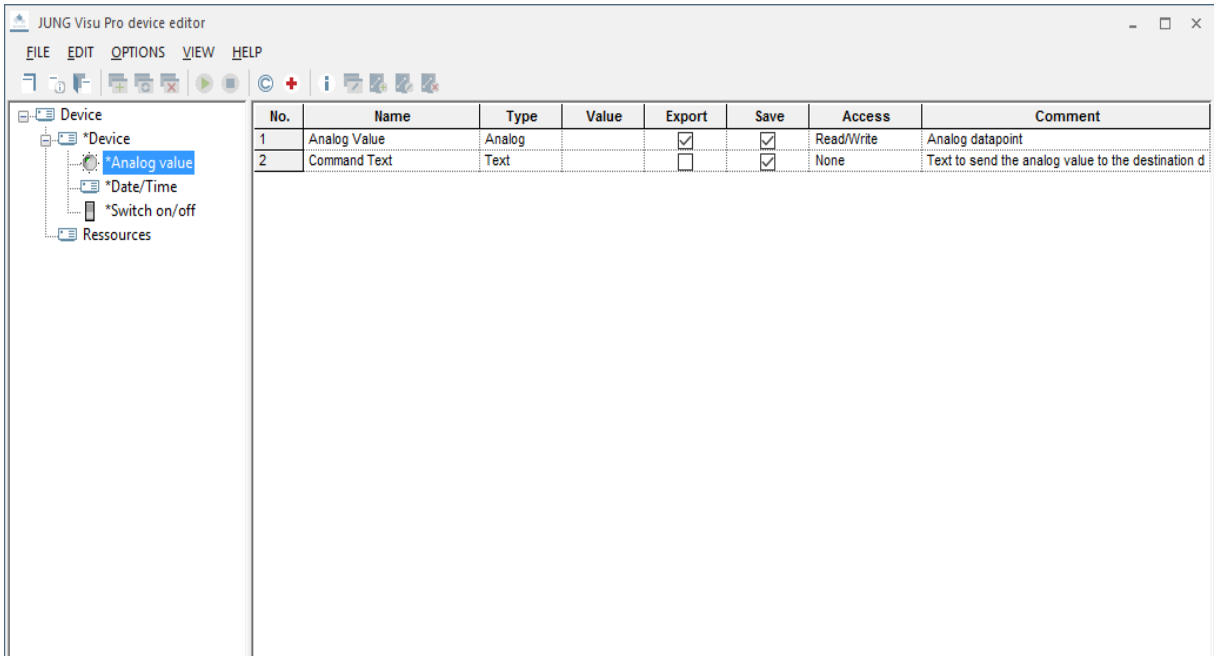


Enter the following information in the dialog:

Name: Enter a new name for the process interface.

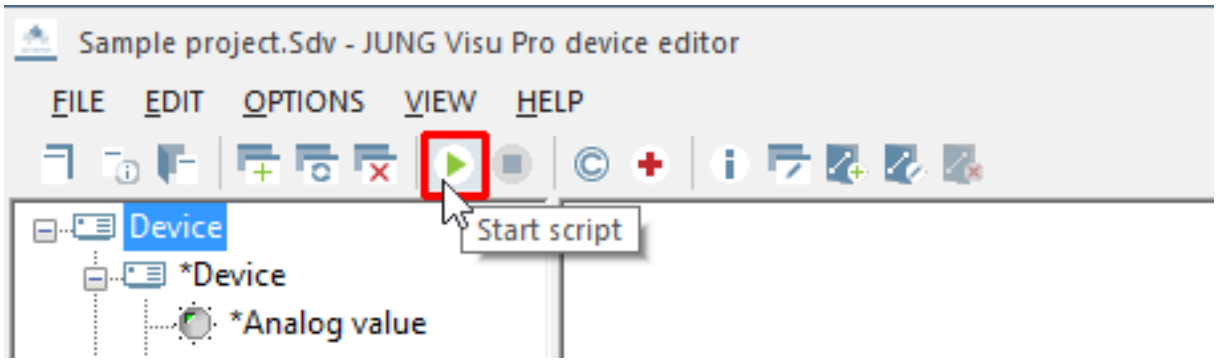
Comment: The specification of a comment is optional.

Click the button ***Create*** to submit the entries.



4.2.2 Start and Stop Process Interface

Start Process Interface



To start a process interface:

- select the process interface, click the right mouse button and select in the context menu the command ***Start process interface*** or
- choose the menu item ***Options*** and click ***Start process interface*** or
- click the button in the toolbar.

The script for the selected process interface starts.

Restart Process Interface

To restart a process interface:

- Select the process interface, click the right mouse button and select in the context menu the command ***Restart process interface*** or
- choose the menu item ***Options*** and click ***Restart process interface***.

The script for the selected process interface restarts.

Stop Process Interface

To stop a process interface:

- Select the process interface, click the right mouse button and select in the context menu the command ***Stop process interface*** or
- choose the menu item ***Options*** and click ***Stop process interface*** or
- click the button in the toolbar.

The script for the selected process interface stops.

Automatic Start of Process Interfaces

After a devices project is loaded in the device editor – this can be achieved by automatic start of a project – the device will automatically start a process interface if this behavior has been selected in its process interface properties dialog.

To open this dialog, select the process interface, choose the menu item ***Edit*** and click ***Process interface properties***. Select the checkbox ***Auto connect when project is opened***. Click the button ***OK*** to submit the entry.

4.2.3 Edit Process Interface

The actual content of a process interface is determined by a description of its possible data points and a collection of scripts for its dynamic behavior, for example methods for data transmission and interpretation.

Here we use the process interface type "UDP commands" as an example. This process interface type is installed by the setup program.

The process interface type "UDP commands" allows to send internet datagrams according to the User Datagram Protocol (UDP) standard. This is a standard supported by many IP (Internet Protocol)-capable devices.

With the "UDP commands" process interface type, binary values, analog values and time information can be sent. These data correspond to data points, which have to be defined accordingly.

The data points of each device connected via UDP are grouped into one or more data point folders. There may be an additional folder for additional data, which are not transferred to the device and not described as data points. This is called the resources folder.

Note that data points are visible in the process model, data in the resources folder are only visible in the devices project.

Finally the folders used by a process interface type are determined by the process interface type itself and its usage, not by the device editor. The role of the device editor is to provide the API and a user interface for the process interface type.

The "UDP commands" sample provides the following folders:

- The "Resources" folder
- Here the UDP communication port is defined.
- More data point folders which by default have the name "Device", since they represent commands to a physical device.

To open the context menu of a folder or folder item, click the right mouse button.

Important note regarding help files

To invoke the help file for the device editor, choose the menu item ***Help*** and click ***Content*** or the key ***F1*** or the respective toolbar button. This is the usual way to get help information.

Additionally, since there are one or multiple process interfaces residing in a devices project, a method to get help about a process interface has been integrated.
If there is a help file for a process interface and if the process interface is selected in the tree structure, the methods above will not open the general editor help but a PDF file for the selected process interface.

4.3 Data Point Folders

4.3.1 Create and Delete Data Point Folders

The process interface may allow to define one or more data point folders. In this case the process interface specifies a sort of scheme for data point folders instead of a fixed set of folders. The user can define additional folders according to this scheme.

In the tree view of the devices project these user-defined folders are marked with an asterisk '*'.

Create data point folder

To create a subfolder, select the parent folder. Then choose the menu item ***Edit*** and click ***Create data point folder*** or right-click the parent folder and use its context menu. The dialog ***Create data point folder*** opens. It contains all available subfolders.

Delete data point folder

To delete a user-defined folder, select the folder. Then choose the menu item ***Edit*** and click ***Delete data point folder*** or right-click the folder and use its context menu.

Answer the confirmation prompt with the button ***Yes***.

4.3.2 Rename Data Point Folder

To rename a folder,

- select the folder you want to rename, choose the menu item ***Edit*** and click ***Rename folder*** or
- click the right mouse button and select in the context menu the command ***Rename folder*** or
- click the selected folder once more to open its input field.

Then you can edit the folder name directly.

4.3.3 Edit Data Points

Create/Delete: The individual data points in a data point folder are defined by the respective process interface and cannot be created or deleted by the user.

Rename: The names of data points can be edited. Click twice the name of the data point to open its input field and to edit the name.

Export: The *export* flag decides whether the data point is visible in the process model when the devices project has been imported to the process model editor. The default is predefined by the author of the process interface type.

Save: The *save* flag decides whether the current value of the data point will be saved when the device editor closes and restored when the project is loaded.
The default is predefined by the author of the process interface type.

Access: You can define access rights for each data point.

None	Neither read nor write, still some process interface could update the value.
Read	Only read commands can be sent.
Write	Only write commands can be sent.
Read/Write	Both reading and writing is allowed.

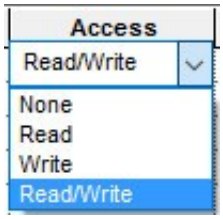
Attention: *These access rights defined by the user will only take effect if they are recognized by the scripts of the process interface type. This is the responsibility of the author of the process interface type!*

Change value: When the process interface is active, the value of a data point can be changed as follows:

1. Click on the field ***Value*** of the data point and enter the value.
2. To send the value, select the row of the data by clicking the ***No.*** column.
3. Click the right mouse button to open the context menu. In the context menu, click the command ***Send value*** to send the current value to the process.

The actual transmission of the value must be implemented in the script for the process interface type.

4.4 Running the "UDP commands" Process Interface



Access control

The script evaluates access rights. UDP datagrams will only be sent when access rights "write" or "read/write" are specified for the datapoint.

To start the "UDP commands" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device has been successfully established, the script will recognize changes to the data points from the side of the process model or the visualization. Then the script checks access rights and builds the strings to be sent. Finally, the script transfers these telegrams to the devices editor which actually performs the sending.

Execution of Write Commands

Important: After the script has transferred the UDP datagrams to the devices editor, the execution of the command will be reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device!
The new value sent will be reported back to the process model if the access rights "Write" or "Read/Write" have been set.

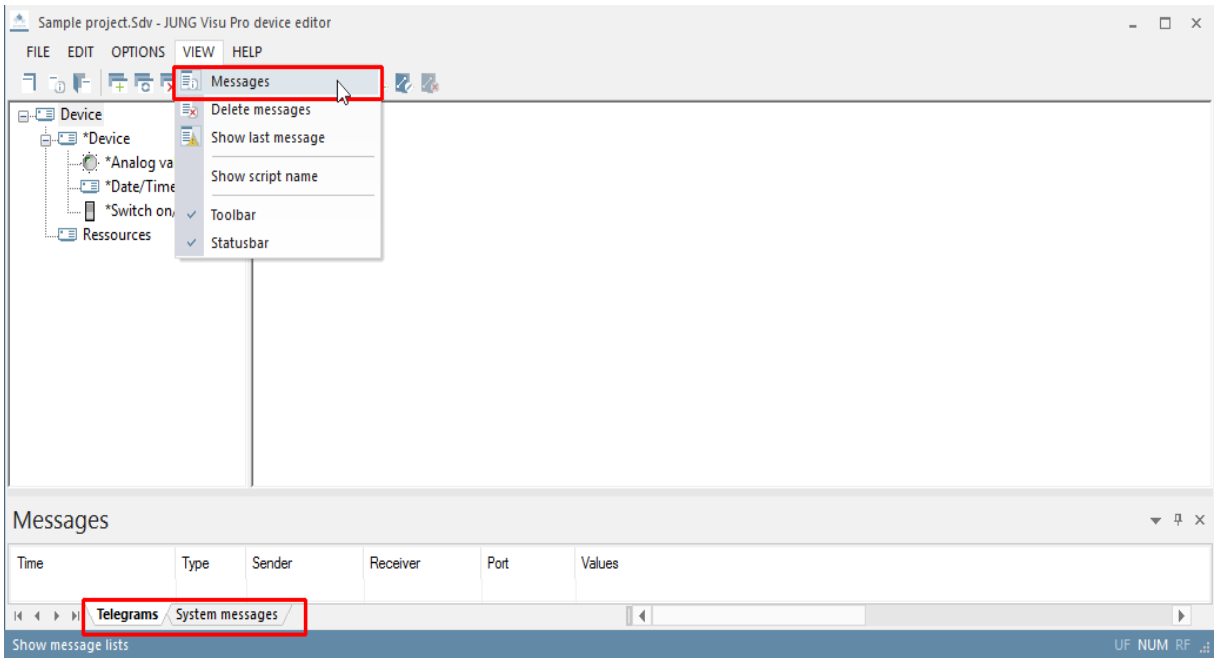
Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no UDP telegram will be created. Instead, the current value of the data point which is the value last sent, will be reported back.

The current value will be reported back only if the access rights "Read" or "Read/Write" have been set.

4.5 Message List

To show or hide the message list, choose the menu option ***View*** and click ***Messages*** .



The messages derive from two kinds of sources:

- the device editor:
- Messages regarding the operation of the device editor, including all operations regarding the state of process interfaces, e.g. loading, stopping,..
- the individual process interfaces:
- These messages are specific to individual process interfaces. See the descriptions of the individual process interfaces to get more information.

The messages are not saved in the device editor.

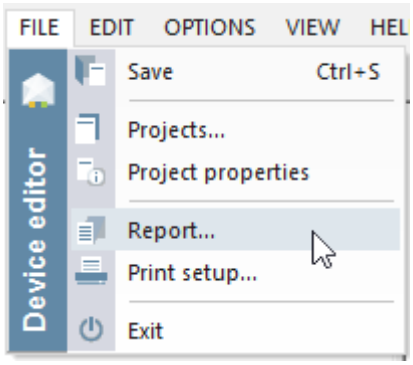
To turn auto scroll on or off, choose the menu item ***View*** and click ***Show last message***. When turned on, the window will always automatically scroll to the last message.

To delete all messages in the message list, choose the menu item ***View*** and click ***Delete Messages***.

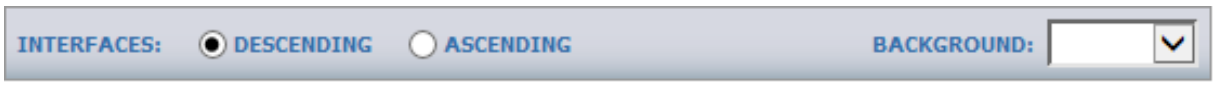
5 Reports

- The device editor creates a report of the devices project with its process interfaces and their properties.
- The report can be printed, along with a print preview.

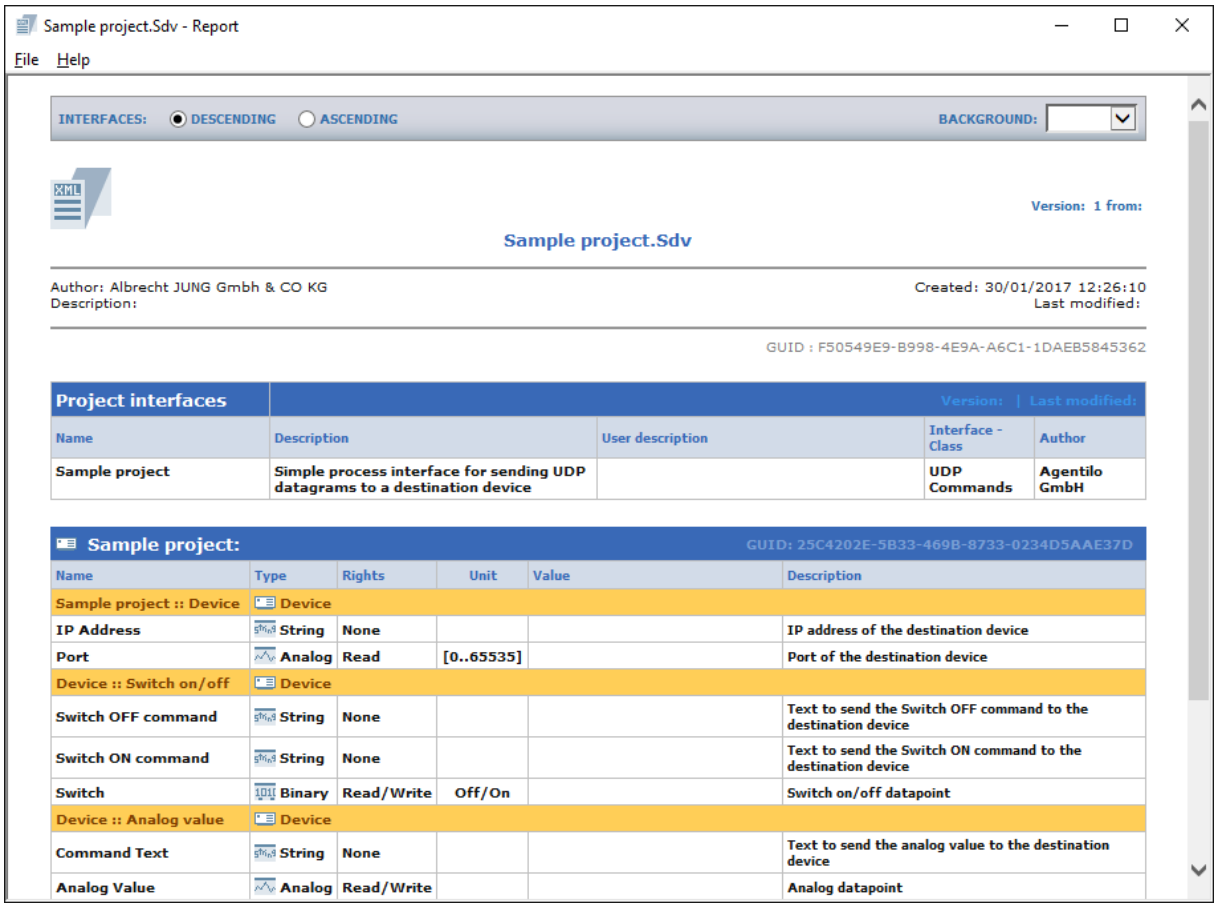
5.1 Create Report



To create a report for the current project, choose the menu item ***File*** and click ***Report***.



The menu at the top of the report window provides an option for sorting the report in descending or ascending order.



5.2 Print Preview

The print preview may be scaled.

Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.

Enlarges the print preview.

Reduces the print preview.

Button to hide the margin markers.

Button to display the margin markers.

With these , markers the user has visual control over the margin setting.

JUNG Visu Pro
Device Editor

Process Interface Assa Abloy

Version 1.0

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Creation of the Data Point Folders....4
   1.1 Data Point Folder Room....5
   1.2 Data Point Folder Common....6
   1.3 Access Control....7
   1.4 Data Point Folder Resources....7
2. Running the Process Interface....8
   2.1 Execution of Write Commands....8
   2.2 Execution of Read Commands....8

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the **Assa Abloy Interface** described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The **Assa Abloy Interface** is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these term, you are not allowed to use the Assa Abloy Interface.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface ***Assa Abloy*** provides data points for Assa Abloy doorsolutions, which receive states via UDP. It is possible to create multiple rooms in an interface. For each room the interface will document, if staff- or guestcards are used to enter the room.

The values will be reset daily. It is possible to enable a protocol, which will be saved in the CSV format. If activated, the necessary information of a room will be written in this protocol.

1. Creation of the Data Point Folders

After creating the process interface, a default data point folder ***Floor*** is shown in the data tree.

A ***Floor*** folder can contain any number of ***Room*** folders. To create a new floor, click with the right key of your mouse on the process interface and select „Create a new data point folder". The same can be done to create new rooms, by right clicking on a floor.

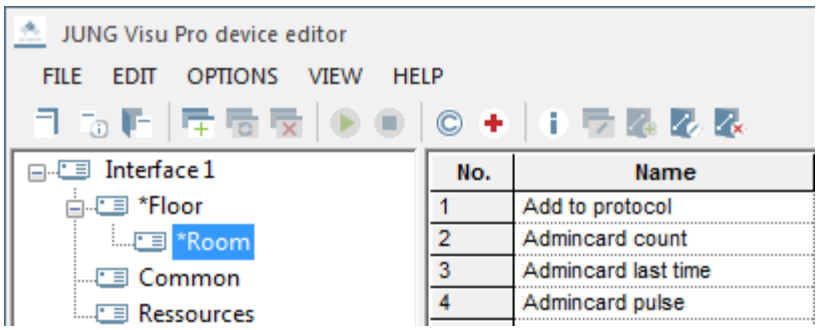
Important:
Every room in the process interface needs to contain a valid **room number**. This number has to be **unique**. If no room number is entered or if a number is used twice the interface will throw a configuration error!

Alongside the floors exists another data point folder named ***Common***. This folder contains variables, which are necessary for the interface to work.

The ***Resources*** folder contains the communication port which is monitored to receive new messages.

The data point folders ***Floor*** and ***Room*** as well as the ***Common*** folder can be renamed. The name of the folders do not matter for the evaluation of the script.

Every folder ***Room*** contains data points, which monitor the usage of the door mechanism.



1.1 Data Point Folder Room

Alongside the binary data points, which symbolize if a staff or guestcard is used, there are a view more. Full list of all data points in a room:

Add to protocol:
Determines, if the values of this room should be written into the protocol.
If left empty, it will be set to 0 on initialization.

Admincard count:
Counts the daily usage of the admincard.
If left empty, it will be set to 0 on initialization.

Admincard last time:
Timestamp from the last use of the admincard.

Admincard pulse:
The pulse duration can be set with the **Pulse duration** data point in the ***Common*** data point folder. Will be set to 1 if the admincard is used. After the pulse duration it will be set to 0 again.
If left empty, it will be set to 0 on initialization.

Admincard used:
Will be set to 1, if the admincard is used on the current day.
If left empty, it will be set to 0 on initialization.

Both cards used:
Will be set to 1, if the admincard and the guestcard are used on the current day.
If left empty, it will be set to 0 on initialization.

Door locked:
Will be set to 1, if the door of the room is locked.

Guestcard count:
Counts the daily usage of the guestcard.
If left empty, it will be set to 0 on initialization.

Guestcard last time:
Timestamp from the last use of the guestcard.

Guestcard pulse:
The pulse duration can be set with the **Pulse duration** data point in the ***Common*** data point folder. Will be set to 1 if the guestcard is used. After the pulse duration it will be set to 0 again.
If left empty, it will be set to 0 on initialization.

Guestcard used:
Will be set to 1, if the guestcard is used on the current day.
If left empty, it will be set to 0 on initialization.

Roomname:
Name of the room.

Roomnumber:
Number of the room. Has to be unique!

1.2 Data Point Folder Common

The data point folder ***Common*** contains following data points:

Configuration error:
Will be set to 1, if an error in the configuration occurs.

Enable protocol:
Enables/Disables the creation of the protocol.
If left empty, it will be set to 0 on initialization.

IP address:
The IP address of the JUNG Gateway.
Has to be set, if **Use IP address as UDP filter** is used.

Protocol path:
The storage location of the protocol file without a file name. Has to be the path without a filename like C:\Temp\
ATTENTION: Access rights to the given folder is necessary!
Has to be set, if **Enable protocol** is used.

Protocol time hour:
Together with the **Protocol time minute** data point, this results in the time at which the rooms are reset and, if necessary, the protocol is written.
If left empty, it will be set to 0 on initialization.

Protocol time minute:
Together with the **Protocol time hour** data point, this results in the time at which the rooms are reset and, if necessary, the protocol is written.
If left empty, it will be set to 0 on initialization.

Pulse duration:
Duration in seconds of an active pulse in a room.
If left empty, it will be set to 1 on initialization.

Runtime error:
Will be set to 1, if an error occurs at runtime.

Sending port:
Port of the JUNG Assa Abloy Gateway-Application to receive commands from the interface.

Text separator:
Specifies the character that is used to separate text in the report.
If left empty, it will be set to '' on initialization.

Value separator:
Specifies the character that is used to separate columns in the report.
If left empty, it will be set to ; on initialization.

Use IP address as UDP filter:
If enabled, the interface will only evaluate UDP messages which are send from the given IP address.
If left empty, it will be set to 1 on initialization.

Log path:

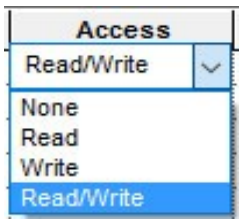
The storage location of the log file without a file name. For example: C:\Temp\
Has to be set, if *enable logging* is enabled.
ATTENTION: Access rights to the given folder is necessary!

Activate/ Deactivate logging:

Enables or disables logging to a specified file.
If left empty, it will be set to 0 on initialization.

Important:
The interface will only work, if **all room numbers** as well as the **protocol time** are set.
If the protocol is enabled, the **protocol path** has to be set, too.
If **Use IP address as UDP filter** is used, the interface will not start as long as no IP address is entered.
In this cases the data point **Configuration error** will be set to **1**.

1.3 Access Control



The script evaluates access rights. Datagrams will only be sent when access rights **write** or **read/write** are specified for the data point. You should not change the access rights in this script.

1.4 Data Point Folder Resources

The folder Resources contains the address configuration for the communication port. The telegrams will be received on this port (default: 5702).

2. Running the Process Interface

The process interface can be started with **Menu – Options – Start process interface** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

If all parameters are set correctly, the interface will wait for incoming UDP commands and evaluate them. At the configured time all values will be reset daily. Before that they will be written into the protocol, if enabled.

2.1 Execution of Write Commands

Important: Transferring the datagrams to the devices editor is the last operation of the script regarding sending. Afterward execution of the command is reported as successful to the process model. There is no check whether the data could be sent and have been received by the device!

With access rights **write** or **read/write** the new value sent will be reported back to the process model, otherwise not.

2.2 Execution of Read Commands

When a read command is invoked by the process model respectively the user, the actual value of the data point, which is the last send value, will be reported back.

The actual value will be reported back only if the access rights **read** or **read/write** have been set.

JUNG Visu Pro
Device Editor

Process Interface Daylight Simulation

Version 1.0

Content:

DISCLAIMER....1

FUNCTIONALITY....3

1. Creation of a Daylight Simulation....4
   1.1. The Data Point Folder Daylight Simulation....4
   1.2. The Input Data Points of the Daylight Simulation....4
   1.3. The Output Data Points of the Daylight Simulation....5
      1.3.1 Configure the Access Rights....6
   1.4 Data Point Folder Resources....6
2. Running the Process Interface....7
   2.1 Execution of Write Commands....7
   2.2 Execution of Read Commands....7

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the ***Daylight Simulation Interface*** described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence. The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of wilfully misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer. The ***Daylight Simulation Interface*** is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these term, you are not allowed to use the Daylight Simulation Interface.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® und ETS® are registered trademarks of the KNX Association.
LON® is a registered trademarks of the Echelon Corporation registered in the United States and other countries..
OPC® is a registered trademarks of the OPC Foundation.
Sax Basic Engine is a registered trademarks of the Sax Software Corporation.
Adobe Acrobat® is a registered trademarks of the Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of the Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface ***Daylight Simulation*** is calculating the colour temperature of the daylight depending of a time.

The calculation of the new colour temperature value is done by sending a new time to the ***Daylight Simulation*** interface script. The new calculated value is limited by the minimum and maximum value for the colour temperature for the daylight. The colour temperature value is also reachable from outside like the minimum and maximum values. The calculation of the colour temperature is done in Kelvin.
In addition to the colour temperature value for the daylight a percentage and a scale value between 0 and 255 will be calculated also.

The script is controlled by a start/stop data point which will start and complete the processing of the calculation. A stopped script will not calculate any colour temperature value. If the script is stopped and activated again during a pause the pause will be completed. Will the script activated, there is no automatically calculation of the colour temperature of the daylight. Only by sending a time a new calculation will be processed and a new colour temperature value of the daylight will be provided.

A paused script will not do any more calculations. If a date time value is received during a pause, the received value will be stored at the *time* field. If the pause has finished, a new calculation will be processed based on the last received time and the new colour temperature value will be provided.

There are three ways to pause the script:

- Sending a pause request as binary value
- Sending a new colour temperature value for the daylight
- Sending a scene selection

All these kind of pauses can be finished by disabling the pause or by enabling the script.

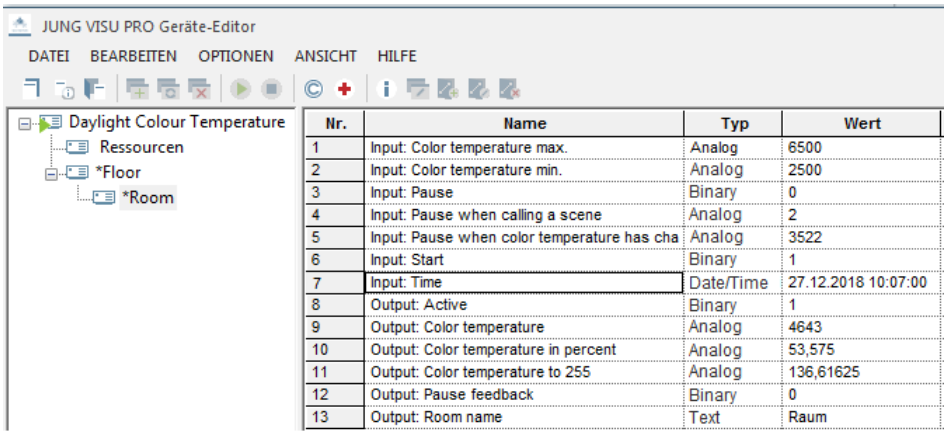
1. Creation of a Daylight Simulation

After a new process interface has been created, a new default data point folder is shown in the tree. Its name is ***Room***. This folder is placed in a data point folder ***Floor***.

The folder ***Room*** containing all necessary data points for the script to do the calculation of the colour temperature of the daylight.

It is possible to create multiple ***Room*** and ***Floor*** folders. Use the right click at the parent folder with the name of the process interface to open a context menu. Within the context menu you can create several new folders for the new floors. Inside a ***floor*** you can create multiple new rooms by right click at the folder ***floor***.

You can rename the data point folders ***Room*** and ***Floor*** to specify a better description for the place of action. The name of the data point folder will not have any affection to the processing of the script.



Nr.	Name	Typ	Wert
1	Input: Color temperature max.	Analog	6500
2	Input: Color temperature min.	Analog	2500
3	Input: Pause	Binary	0
4	Input: Pause when calling a scene	Analog	2
5	Input: Pause when color temperature has cha	Analog	3522
6	Input: Start	Binary	1
7	Input: Time	Date/Time	27.12.2018 10:07:00
8	Output: Active	Binary	1
9	Output: Color temperature	Analog	4643
10	Output: Color temperature in percent	Analog	53,575
11	Output: Color temperature to 255	Analog	136,61625
12	Output: Pause feedback	Binary	0
13	Output: Room name	Text	Raum

1.1. The Data Point Folder Daylight Simulation

To be able to calculate the value of the daylight colour temperature, it is necessary to know the time of the day the colour temperature is calculated for. Moreover, the script can be controlled by the data points start, stop and pause. The calculated colour temperature value is expressed in Kelvin, as a percentage and as a scaled value between 0 and 255. In summary, the data point folder ***Room*** contains 13 data points.

You can use seven of these data points as input data points. The other ones are used for the output of the colour temperature values and for the feedback of the state of the script.

1.2. The Input Data Points of the Daylight Simulation

Input: Pause

Will pause the ***Daylight Simulation*** script by sending a "1" to this data point. Sending a "0" will continue the script. During the script is paused, no values will be calculated but a time value that is send during the pause, will be send back as feedback. If the pause has finished, a new daylight colour temperature value, based of the last time that was given, is calculated.

Input: Pause when colour temperature has changed

If the colour temperature of the daylight will be changed by the user, the script will pause. The information about the new colour temperature value has to be written in this data point field. The information does not need to be the new colour temperature value it self.

Input: Pause when calling a scene

If a scene call is made while the script is active, the script will be paused. The information about a new scene call is made has to be written in this data point field. The written value has not to be the number of the called scene itself.

Input: Start

Sending a "1" into this data point field, the calculation of the daylight colour temperature will start by the script. The calculated colour temperature value is expressed in Kelvin, as a percentage, and as a scaled value between 0 and 255.

Input: Colour temperature max.

The limitation of the maximum of the calculated daylight colour temperature value is controlled by this input data point field. There will be no value exposed that is higher than the value that is stored in this data point field.

Input: Colour temperature min.

The limitation of the minimum of the calculated daylight colour temperature value is controlled by this input data point field. There will be no value exposed that is lower than the value that is stored in this data point field.

Input: Time

The day time value that is needed to do the calculation of the daylight colour temperature. After sending a new time value, a new calculation will run and a new value will be exposed through the colour temperature output data point.

1.3. The Output Data Points of the Daylight Simulation

Output: Active

Containing the value one if the script is running, otherwise a zero. If the script is active, every sending of a new day time will run a new calculation of the daylight colour temperature values.

Output: Pause feedback

Contains the value one, if the script was paused. Otherwise a zero is presented.

Output: Room name

This data point contains the name of the room. Changing this name will not have any effect of the calculation of the daylight colour temperature.

Output: Colour temperature

The calculated colour temperature of the daylight in Kelvin. The calculated value represents the colour temperature of the daylight for the time that was send to the input data point *time* before. This calculated value is limited by the input data point values of the fields "Input: Colour temperature max." and "Input: Colour temperature min.". If the script is paused, no new colour value will be calculated. If the script continues after a pause, the last colour value will be delivered.

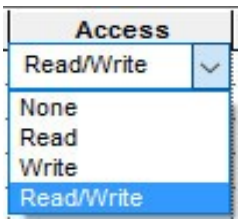
Output: Colour temperature in percent

This value represents a percentage value related to the colour spectrum that is limited by the minimum and maximum input data points.

Output: Colour temperature to 255

This field contains the value of the daylight colour temperature scale to a range from 0 to 255.

1.3.1 Configure the Access Rights



The script evaluates the access rights settings. Values will only be send if the access rights settings are set to **write** or **read/write**. All settings have been made optimally for this script. You should not change these values.

1.4 Data Point Folder Resources

This folder containing no data.

2. Running the Process Interface

The process interface script can be started automatically by the process model if configured or manually by the user with a right click at the ***Daylightsimulation*** folder and the menu option ***Start process interface***.

Now the script is ready to calculate new daylight colour temperature values depending on the time that was send to the time data point.

2.1 Execution of Write Commands

Important: Transferring the datagrams to the devices editor is the last operation of the script regarding sending. Afterwards the execution of the command is reported to the process model as successful. There is no check whether the data could be sent and have been received!

The access rights have to be set to ***write*** or to ***read/write*** to receive any feedback of the current value.

2.2 Execution of Read Commands

If the process model or the user performed manually a ***value read*** command the current value of the data point will be send back as result of the reading operation.

The settings of the access rights will be evaluated by the script. Therefore the access rights have to be set to ***read*** or to ***read/write*** to receive any feedback of the current value.

How to –

Daylight simulation – Human Centric Light (HCL)

How to create a daylight simulation with JUNG Visu Pro and JUNG KNX DALI gateway TW

Light and emotions always go hand in hand – the proper lighting in a building is crucial for people's well-being, health and productivity.

Human Centric Light (HCL) enhances modern lighting design with the biological lighting effect. HCL puts people at the centre of planning and incorporates natural biorhythms with natural lighting similar to daylight and a melanoptic lighting effect. In addition to the correct illuminance [lux], the right colour temperature [Kelvin] is also essential.

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
58579 Schalksmühle, Germany

Telephone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89
kundencenter@jung.de
www.jung.de

Technical consultation
KNX
+49 (0)2355 /80 6556

Required products

	Device	Ref.-no.
	JUNG Visu Pro Server	JVP-SERVER-H2
	JUNG KNX DALI gateway TW	2099REGHE
	JUNG KNX IP interface	IPS300SREG

Contents

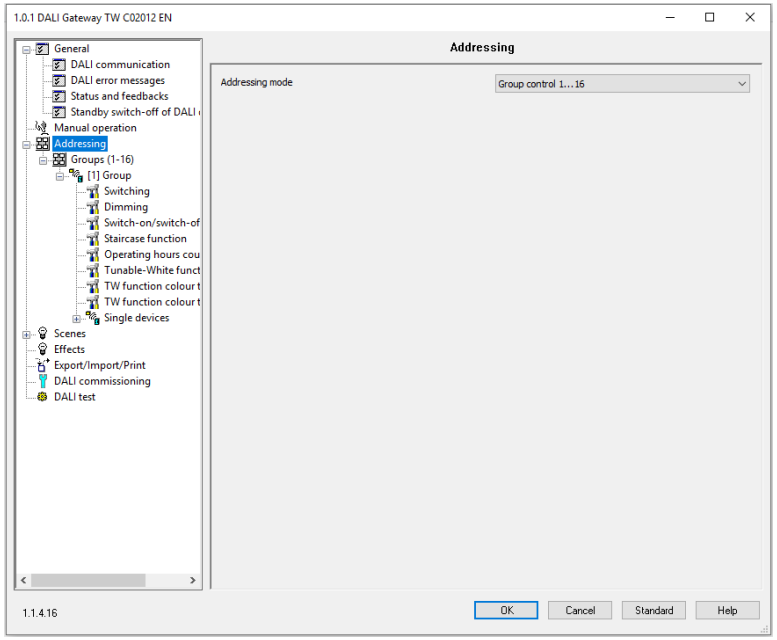
1. JUNG KNX DALI GATEWAY 4
1.1 Configuring the KNX DALI GATEWAY....5
1.1.2 Groups and single devices 6
1.1.3 Setting parameters for groups 7
1.1.4 Tunable-White function 8
1.1.5 TW function colour temperature 1 9
1.1.6 Scenes....10
1.1.7 DALI commissioning....11
1.1.8 Communication objects....12
2. JUNG Visu Pro....13
2.1 Configuring JUNG Visu Pro 14
2.2 Device editor....15
2.2.1 Data point folder 16
2.2.2 Data points of the simulation 17
2.3 KNX editor....18
2.3.1 KNX settings....19
2.4 Process models editor 20
2.4.1 Data points....21
2.4.2 Forwarding process model 22
2.5 Visualization editor....23
2.6 Starter....24

1. JUNG KNX DALI GATEWAY

- Control of max. 64 DALI devices in max. 32 groups
- Setting the colour temperature for luminaires with DALI device
- Type 8 for Tunable White in accordance with IEC 62386-209
- Suitable for operation in emergency lighting systems
- Individual, group or central addressing
- 16 light scenes
- Effect control for dynamic lighting effects and colour plays
- Online catalogue

1.1 Configuring the KNX DALI GATEWAY

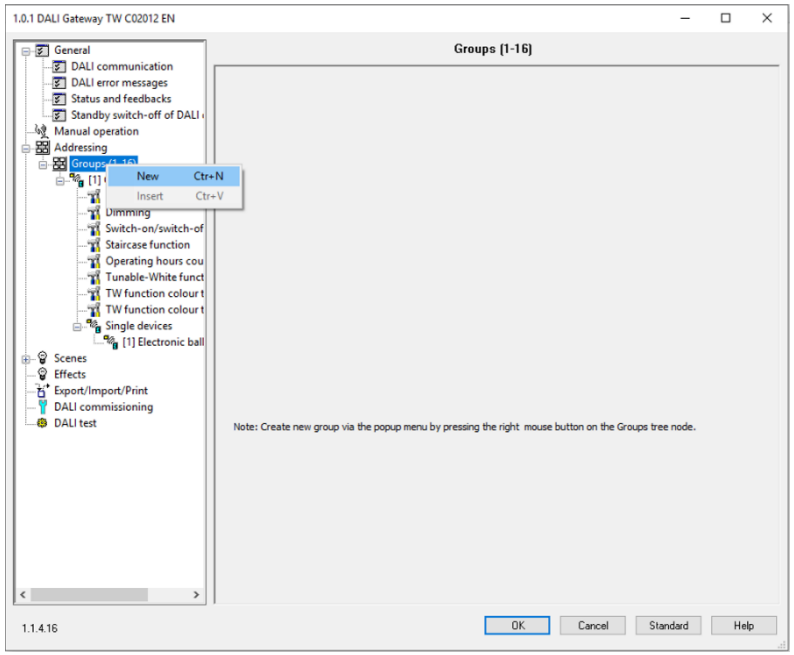
ETS product-specific parameter dialogue



- Import the product data base of the JUNG KNX DALI GATEWAY TW via the catalogue into your ETS.
- Add the device to your project structure and open the product-specific parameter dialogue under "**Parameters**".
- Under "**General**" you can set the communication behaviour and the settings for the manual operation of the device – do not change the default settings for now.
- Go to the "**Addressing**" tab in the tree node and define the type of addressing for your DALI luminaires:
  - E.g. Group control 1...16
    (Control of up to 16 DALI groups)

1.1.2 Groups and single devices

ETS product-specific parameter dialogue

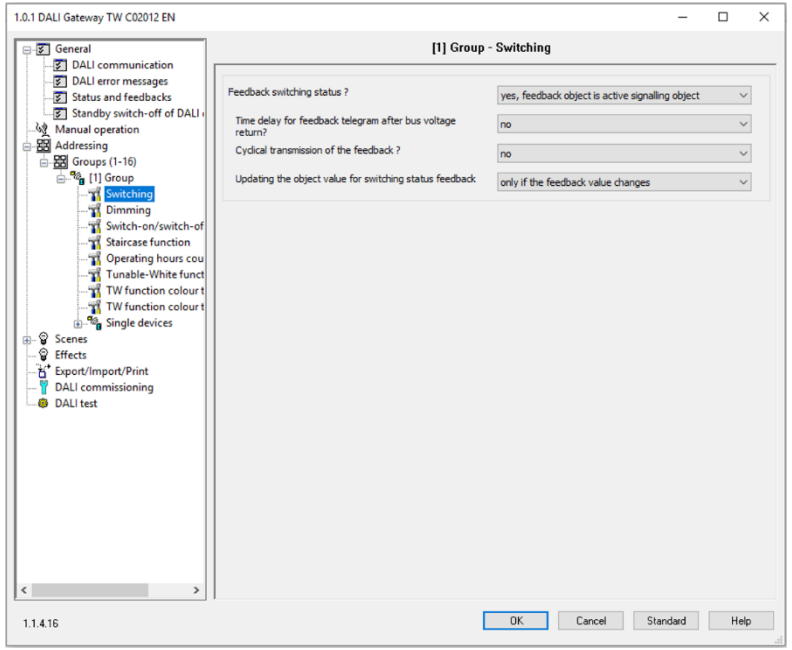


- The node “Groups (1-16)” appears in the tree node.
- Press the right mouse button on the “Groups (1-16)” tree node and a popup menu will appear – here you can create new DALI groups.
- Create a new DALI group, e.g: **[1] Group**
- Press the right mouse button on the “**Single devices**” tree node to add up to 64 individual devices (EVG) to the group.
- Now add **one single device** ([1] EVG) to the group
- Click on the **Single device** ([1] EVG) and configure the device type and the short address for the single device;

  For the Tunable-White function, select the type “**Ballast for Colour-Control (Tunable White)**” or “**Multitype**”
- The following settings are used in this example:
  - Name: **EVG**
  - Short address (1...64): **1**
  - Device type: **Multitype**

1.1.3 Setting parameters for groups

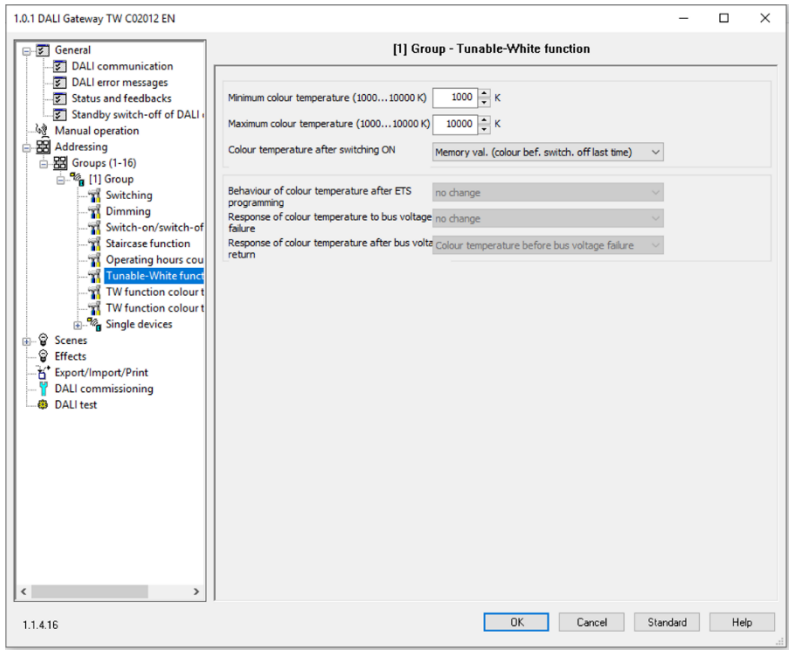
ETS product-specific parameter dialogue



- In the following dialogue you can set the parameters for your DALI groups.
- Set one **active feedback object** each for the **Switching** and **Dimming** functions, as well as for the **switching status** and **brightness.**
- For the DALI characteristic, select the “**linear**” dimming behaviour.
- For the remaining parameters, use the default settings for now.
- Now go to the “**Tunable-White function**”.

1.1.4 Tunable-White function

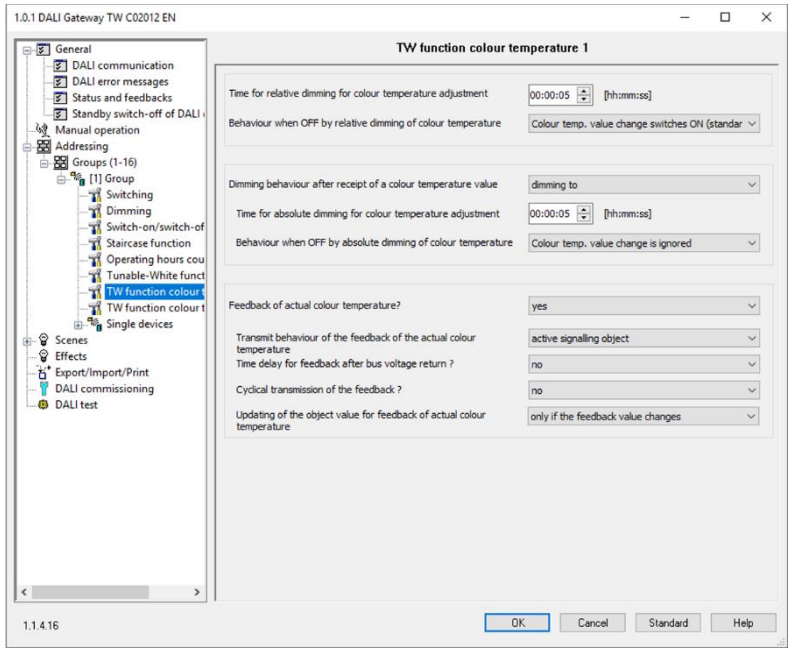
ETS product-specific parameter dialogue



- Set the limits for the minimum and maximum colour temperature:
  - Minimum colour temperature: **1000 Kelvin**
  - Maximum colour temperature: **10000 Kelvin**
- Then set the behaviour of the colour temperature after switching on:
  - Memory val. (colour bef. switch. off last time)
- Go to the "**TW function colour temperature 1**" node.

1.1.5 TW function colour temperature 1

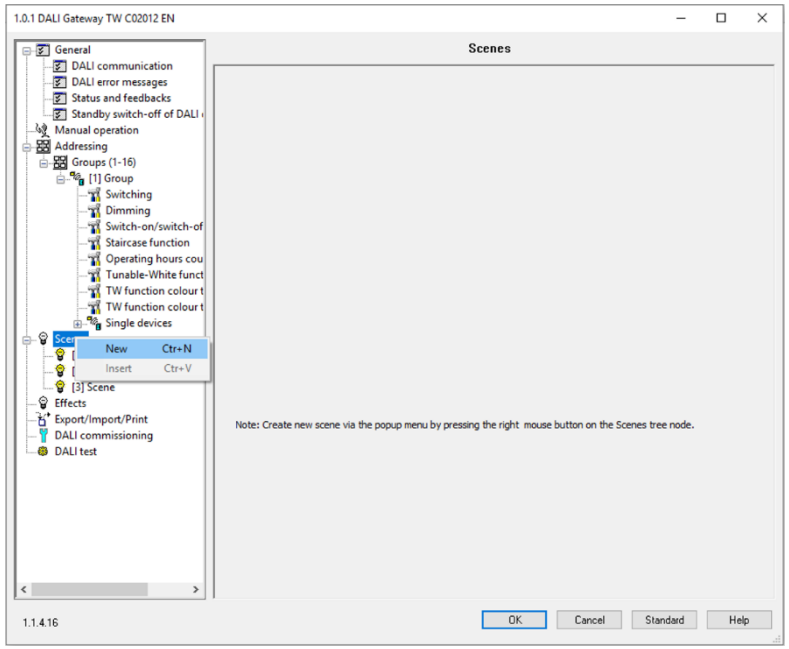
ETS product-specific parameter dialogue



- Set the parameters as follows:
- Behaviour when OFF by absolute dimming of colour temperature
  - **Colour temp. value change switches ON (standard)**
- Dimming behaviour after receipt of a colour temperature value
  - **dimming to**
- Time for absolute dimming for colour temperature adjustment
  - **5 seconds**
  - *The more time you set, the more natural the transitions and colour temperature changes will be!*
- Feedback of actual colour temperature
  - **yes**
- Transmit behaviour of the feedback of the actual colour temperature
  - **active signalling object**
- Use the default settings for the remaining parameters

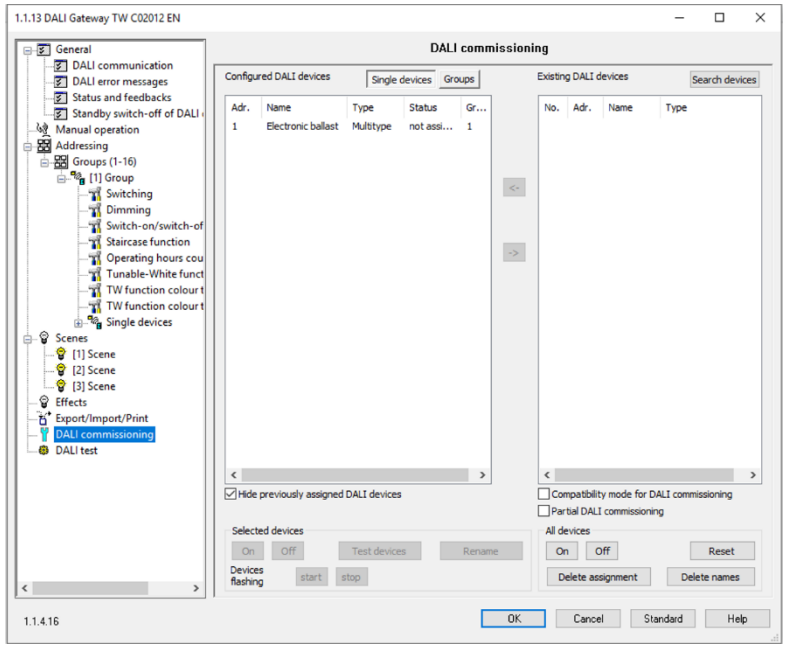
1.1.6 Scenes

ETS product-specific parameter dialogue



- In the "**Scenes**" node, you can create up to 64 scenes for your DALI groups.
- To do this, press the right mouse button on the respective node and create a new scene with "**New**".
- Next, you can e.g. set a defined brightness **[%]** and a colour temperature **[Kelvin]** for your DALI groups when calling up scenes.

1.1.7 DALI commissioning



- You can use the "**DALI commissioning**" node to search for further DALI devices and add them to your installation.

- After entering the settings, close the parameter dialogue with "**OK**" and return to the ETS view.

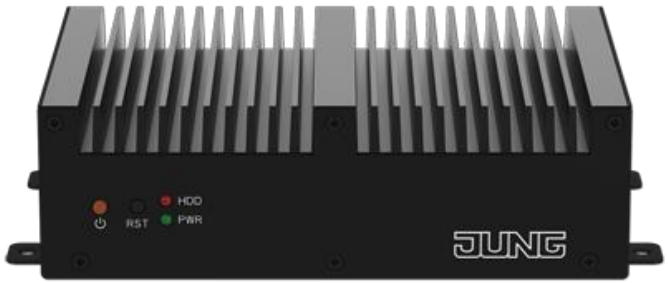
1.1.8 Communication objects

Number	Name	Object Function	Length	Data Type
1	[1] Group	Switching	1 bit	switch
2	[1] Group	Dimming	4 bit	dimming control
3	[1] Group	Brightness value	1 byte	percentage (0..100%)
4	[1] Group	Feedback brightness value	1 byte	percentage (0..100%)
5	[1] Group	Feedback switching status	1 bit	switch
1505	Scenes	Extension input	1 byte	counter pulses (0..255)
1547	[1] Group	Absolute colour temperature(K)	2 bytes	absolute colour temperature (K)
1548	[1] Group	Relative colour temperature	4 bit	dimming control
1549	[1] Group	FB absolute colour temper.(K)	2 bytes	absolute colour temperature (K)

- In ETS you should now see the following **communication objects** for the JUNG KNX DALI gateway TW (left screenshot).
- Connect the objects with the corresponding group addresses from your installation.
- Then export the project file (**.knxproj**).

2. JUNG Visu Pro

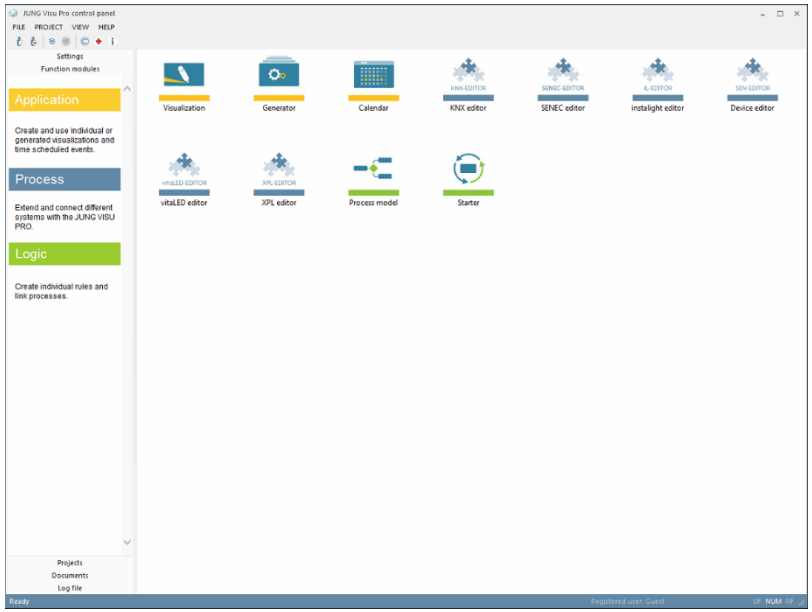
JVP Server



- Visualising and operating system statuses and information on building automation in combination with JUNG Visu Pro software
- Fanless mini PC
- Windows 10, pre-installed
- The visualisation can be realized with end devices with a browser suitable for HTML5 (Chrome, Firefox, Safari)
- Access from max. 10 different clients possible
- Online catalogue

2.1 Configuring JUNG Visu Pro

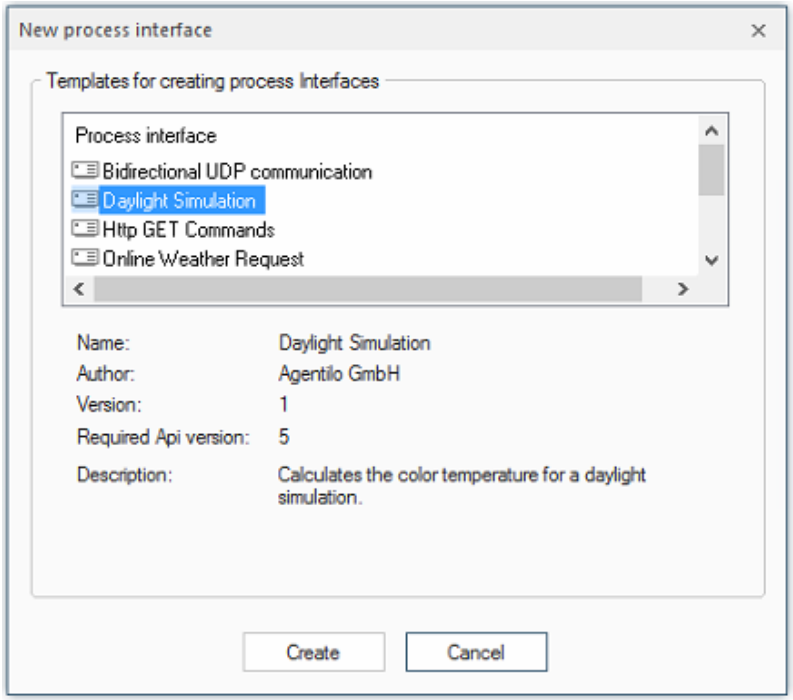
JVP software



- Start JUNG Visu Pro and open the **device editor**.
- Next, create a new device project.

2.2 Device editor

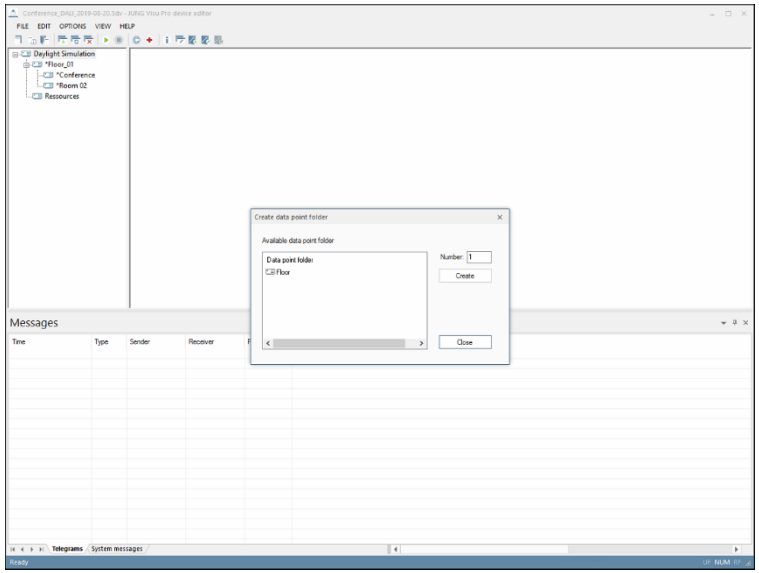
Creating a process interface



- In the **device editor**, create a new daylight simulation under the **EDIT** tab with "**Create process interface**".
- Enter a name for the process interface and click on "**Create**".

2.2.1 Data point folder

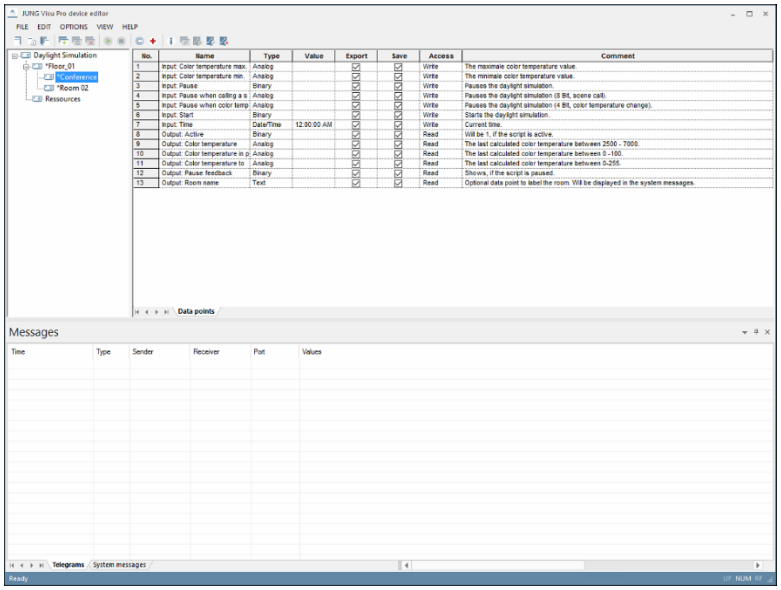
Configuring the daylight simulation



- Press the right mouse button on the process interface ("**Daylight Simulation**") to create a new data point folder for your floor (e.g. **Floor_01**).
- A new floor is added to the tree node.
- Then press the right mouse button on the respective floor and create a new data point folder for a room, e.g. **Conference**.
- An additional room within the first floor is added to the tree node – you can thus create several unrelated daylight simulations for your floor and rooms.

2.2.2 Data points of the simulation

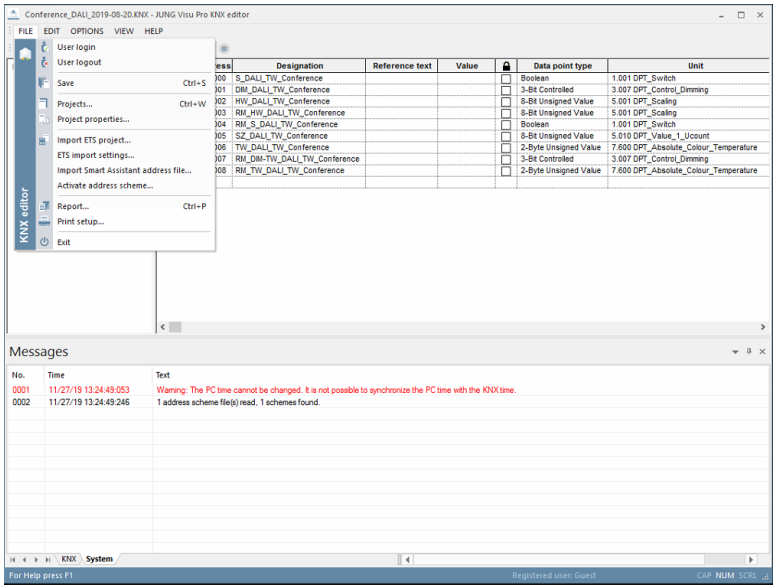
Daylight simulation process connection



- As soon as you have created the daylight simulation including data point folder for a room, the individual data points appear in the list (**right**).
- The **Type**, **Value**, **Access** and a function description (**Comment**) are listed for each data point.
- The Telegrams and Messages tabs can be seen at the bottom.
- Save your configuration!
- Then go to the **KNX editor** and create a new KNX project.

2.3 KNX editor

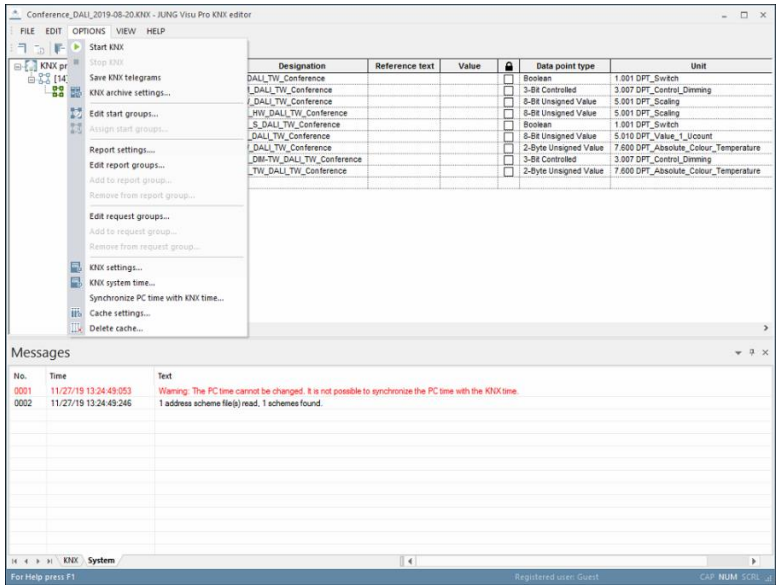
Importing the group addresses



- Import the group addresses or your ETS project (**.knxproj**) via the **FILE** tab in your KNX project with "**Import ETS project...**".
- The connected group addresses of the JUNG KNX DALI gateway TW appear in the tree node of the KNX editor.
- The **Data point type**, **Value** and **Unit** as well as the respective authorisation are listed for each group address.
- *Observe the correct units for the data point types!*

2.3.1 KNX settings

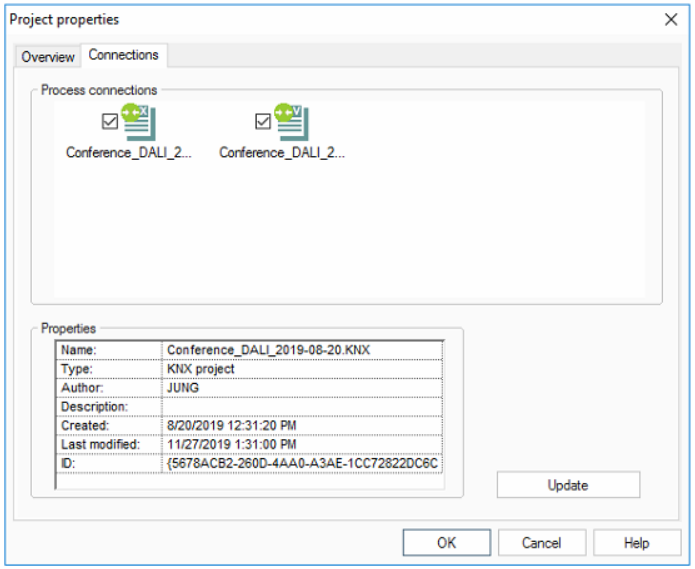
Connection manager



- Now go to the **OPTIONS** tab.
- Under KNX settings configure the settings for the KNX project – start the **connection manager**.
- Set up your standard KNX interface (e.g. **KNXnet/IP** or **USB**) at this point.
- Save your configuration!
- Then open the **process models editor** in JUNG Visu Pro.

2.4 Process models editor

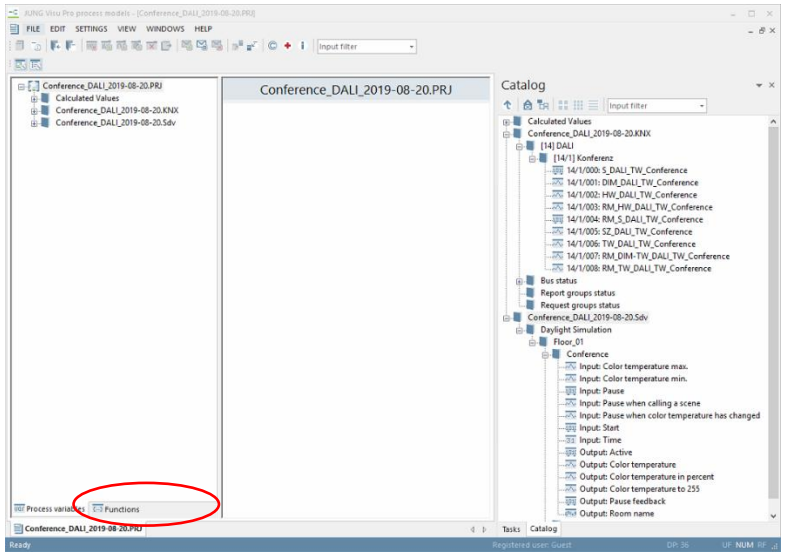
Establishing the process connections



- Create a new **process model**.
- In your **process model**, open the **FILE** tab and then the **Project properties**.
- Under **Connections**, connect the process connection for the daylight simulation (**.SDV**) and the connection for your KNX project (**.KNX**) to the process model.

2.4.1 Data points

Data points of the process connections in the process model

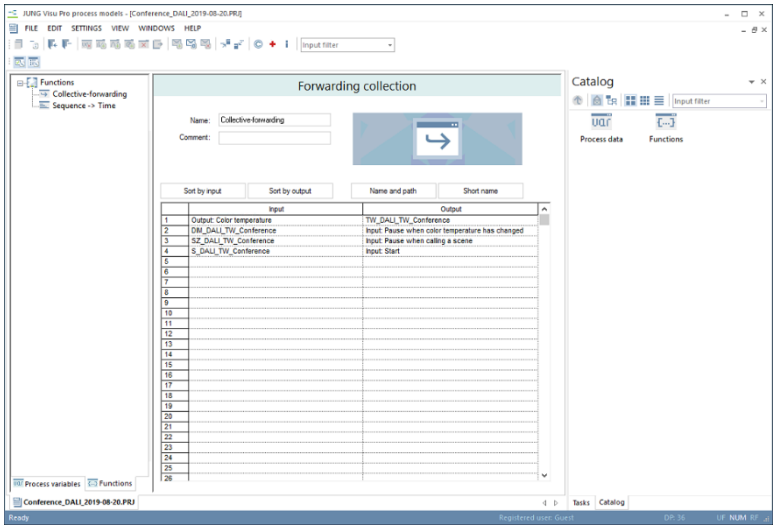


- As soon as you have connected the process connections to the process model, the corresponding group addresses and data points are listed in the tree node (**left**) and in the Catalog (**right**).

- Go to the **Functions** tab.

2.4.2 Forwarding process model

Connecting the logic



- In the **Functions** tab, connect your group addresses with the daylight simulation using the logic of the process model.

- Then, use the **Collective-forwarding** function and connect the following objects:

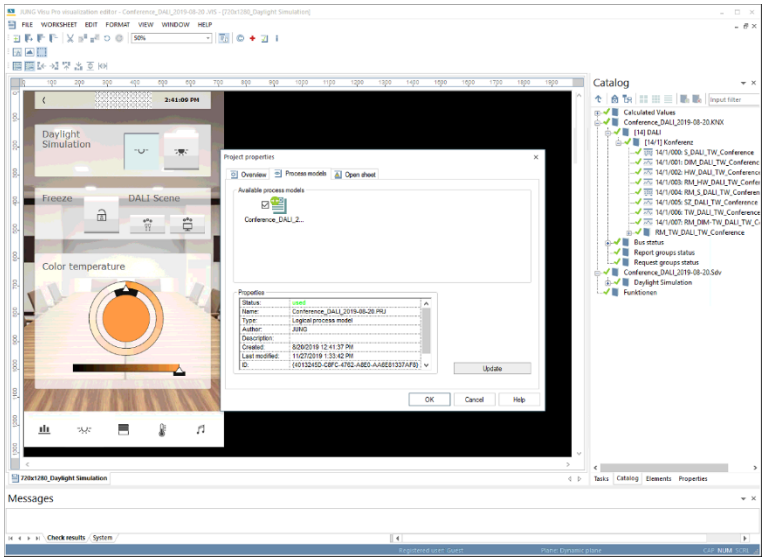
Source of the value	Forward value to
Output: Color temperature	TW_DALI (2 bytes; DPT 7600)
DIM_DALI (4 bit; DPT 3007)	**Input**: Pause when color temperature has changed
SZ_DALI (8 bit; DPT 5010)	**Input**: Pause when calling a scene
S_DALI (1 bit; DPT 1001)	**Input:** Start

- Save your configuration!

- Go to the **visualization editor**.

2.5 Visualization editor

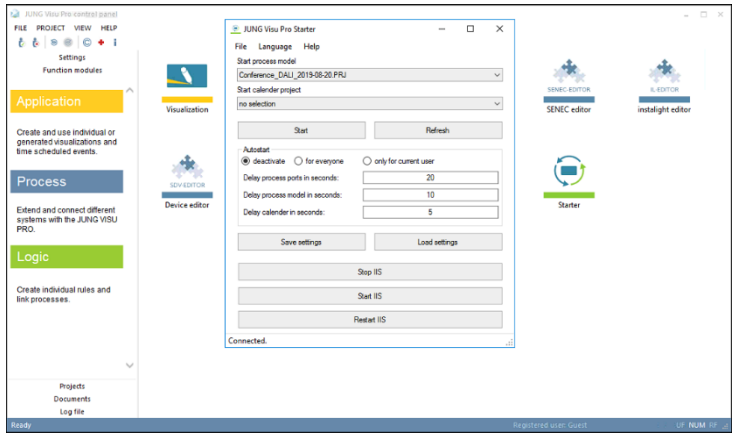
Creating a visualization



- Create a new visualization project in the **visualization editor** or use one of the JUNG templates.
- Connect the respective process model with your visualization project under **FILE** and **Project properties**.
- Then assign the KNX group addresses and data points via drag & drop to the Visu interface.
- Save your configuration!
- Go to the **Starter** function module.

2.6 Starter

Calling up and using the visualization project



- Start your visualization project with the **Starter**.
- Select the process model (**.PRJ**) and click **Start**.
- Open your web browser or the JVP app on your smartphone/tablet and call up the visualization as follows:
- **Via a web browser:**

  http://localhost:8083 or ***http://IP-Adresse-JVP:8083***

- **JVP app:**

  Connection → URL: **IP address JVP:8083**

JUNG Visu Pro
Device Editor

Process Interface “HTTP GET Commands”

Version 1

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Datapoints of an IP-capable Device....4
   1.1. Device Folder....6
   1.2. Creation of Datapoint Folders....6
      1.2.1. Datapoint Folder *Switch on/off*....7
      1.2.2. Datapoint Folder *Analog Value*....8
      1.2.3. Datapoint Folder *Date/Time*....9
      1.2.4. Using Numerical Values in Command Formats....10
      1.2.5. Access Control....10
2. Running the "HTTP GET Commands" Process Interface....11
   2.1 Execution of Write Commands....11
   2.2 Execution of Read Commands....11

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "HTTP GET Commands" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "HTTP GET Commands" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "HTTP GET commands" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface "HTTP GET commands" allows to send HTTP GET requests over the Internet to IP-capable devices, with each an own IP address.

The contents and meaning of the HTTP GET requests depend on the specific device, but the user can specify the contents of the requests in order to implement binary switch, analog set and date/time set commands and corresponding datapoints. These datapoints reside in the datapoint folders.

When using the "HTTP GET commands" software, it may be necessary to configure firewalls, router and other devices to enable the sending of HTTP requests over the network by the software. These measures are the user's responsibility and are not covered here.

1. Datapoints of an IP-capable Device

After a process interface of the "HTTP GET Commands" type has been created, a new tree node with the name ***device*** is added. The node represents an IP-capable device, which can be controlled via sending HTTP GET requests.

From hereafter we will call this node the "device node", though in the actual project it should be renamed to something more descriptive, for example "beamer", "remote control" or "sound system" or "sound system volume" depending on the device and application.

Note that different command/datapoint sets of one device or different devices may be described each in different datapoint folders.

The asterisk in front of the node name signals that more than one node of this kind may be created by right-clicking its tree parent node and opening the context menu.

The HTTP GET request that is sent to a device is composed of the device URL together with a "Query String" that contains the command to the device. The query strings can be configured specifically for each command in the datapoint folders.
The process interface composes the HTTP GET request from the device URL and the respective query string of the datapoint specific command.

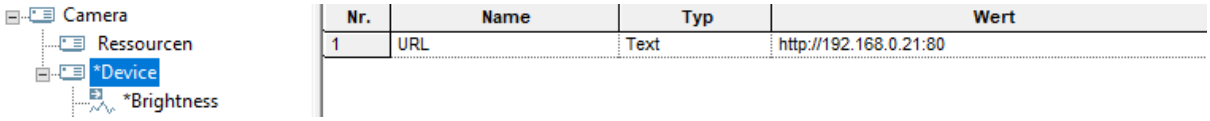
Important: The script does not execute an "URL encoding". The replacement of special characters, that must not occur in a URL, must be performed by the user.

Example for the URL composing by the script:

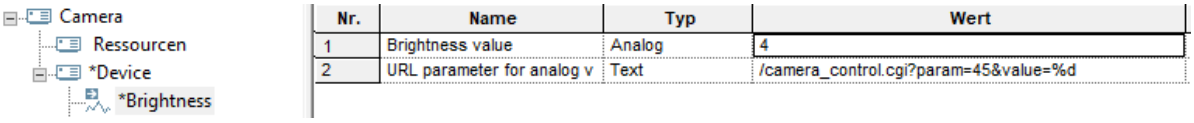
- Command to control the brightness of an IP camera, e.g. <CAMERA IP ADDRESS>/camera_control.cgi?param=45&value=5

- Configuration of the process interface:

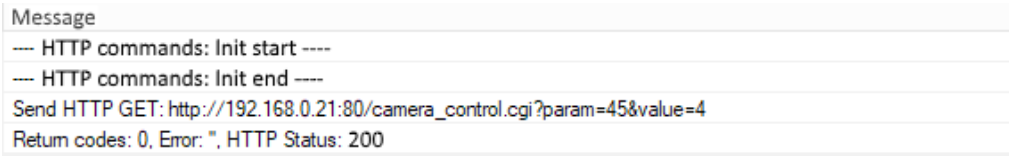
  URL of the camera device, e.g: *http://192.168.0.21:80*



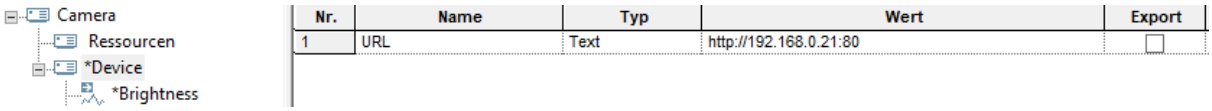
Analog datapoint with the further URL path (e.g. '/camera_control.cgi') and the query string with parameter (e.g. '?param=45&value=<BRIGHTNESS VALUE>') to control the image brightness:



Result: *If the value of the datapoint changes, then the following HTTP GET request will be sent to the camera device:*



1.1. Device Folder



Basic information regarding the device includes its IP address in text form, for example 192.168.0.21:80. If an IP name has been assigned, which resolves to this address, the name can be used instead of the address.

The device folder is created including already the subfolders ***Analog value***, ***Date/Time***, ***Switch on/off*** datapoint folders for possible commands and corresponding datapoints of the respective type. Furthermore the device folder contains the datapoint ***URL*.**

URL: The IP address of the device to be controlled as text,
e.g. *http://192.168.0.21:80*

It is possible to rename the device datapoint folder to give the connected device a meaningful name, e.g. *camera, sound system, remote control,...* The name of the folder does not affect the processing of the script.

It is possible to create more than one device folders.

1.2. Creation of Datapoint Folders

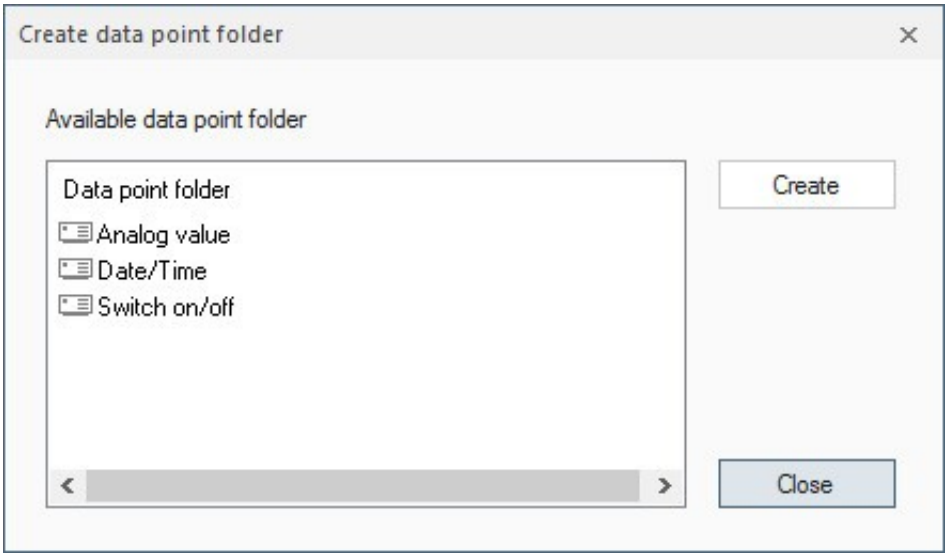
Each datapoint of a device is represented with a datapoint folder inside of the device folder. When the datapoint folder is selected with a left click, a table of the datapoint properties is displayed.

In order to define more datapoints and in addition to the default datapoint folders, more datapoint folders can be created as subitems to the device folder.

To create a new datapoint folder, select the device folder in the structure tree, then click the right mouse button to open its context menu. In the context menu select the command ***Create datapoint folder***. Alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create datapoint folder*** opens. Select the desired datapoint type and click the button ***Create***.

The datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.



1.2.1. Datapoint Folder *Switch on/off*

This folder contains three datapoints:

- **Switch**

  This datapoint is of the type 'binary' and is exported per default to the process model. Exporting means that it will be visible in the process model as a regular binary datapoint. It will be visible in the visualization editor if the process model exports the datapoint as well.

  When the datapoint is switched off, the script will use the text for entry 'URL parameter for OFF' to compose the URL for the HTTP GET request.

  Switching on is similar: the text for entry 'URL parameter for ON' will be used to compose the URL for the HTTP GET request.

- **URL parameter for OFF**

  The value of this variable is the text which will be used to compose the URL to set the logical state 'OFF'.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn off the device.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor! Per default the export flag is not set.

- **URL parameter for ON**

  The value of this variable is the text which will be used to compose the URL to set the logical state 'ON'.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn on the device.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor! Per default the export flag is not set.

1.2.2. Datapoint Folder *Analog Value*

This folder contains two datapoints:

- **Analog Value**

  This datapoint is representing an analog value. It is exported per default to the process model where it is a regular analog datapoint.

  If the value of this datapoint is changed in the visualization, in consequence the script will use the text formatted according to the 'URL parameter for analog value' – entry to compose the URL for the HTTP GET request.

- **URL parameter for analog value**

  The 'URL parameter for analog value' entry defines the command string which will be used to compose the URL to be sent to the device when the value of the analog datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the %d placeholder. Before sending, the script will replace %d with the actual value.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor! Per default the export flag is not set.

1.2.3. Datapoint Folder *Date/Time*

This folder contains two datapoints:

- **Date/Time**

  The type of this datapoint is 'Date/Time'. Per default, it will be exported to the process model.

  If the value of this datapoint is changed in the visualization, in consequence the script will use the text formatted according to the 'URL parameter for date/time value' – entry to compose the URL for the HTTP GET request.

- **URL parameter for date/time value**

  The 'URL parameter for date/time value' – entry defines the command string which will be used to compose the URL to be sent to the device when the value of date/time datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the %s placeholder. Before sending, the script will replace %s with the actual value.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be send.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor! Per default, the export flag is not set.

1.2.4. Using Numerical Values in Command Formats

In command formats it is possible to enter characters as numerical values. A substring (called "escape sequence") in format \ddd, where ddd must be a 1 – 3 digit value between 1 and 255, will be transmitted as a byte with this value.

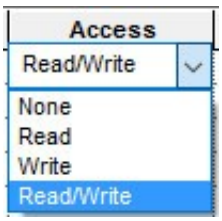
Examples:

\10 encodes a linefeed
\64 encodes the at sign '@'

Note that a byte with the value 0 cannot be used.
If the char '\' is to be inserted literally, it must be duplicated.

Format "\\001" will send "\001" whereas "\001" will send the byte with value 1.

1.2.5. Access Control



The script evaluates access rights. HTTP GET requests will only be sent when access rights ***Write*** or ***Read/Write*** are specified for the datapoint.

2. Running the "HTTP GET Commands" Process Interface

To start the "HTTP GET commands" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device succeeds, the script will recognize changes carried out on the datapoints in the process model respectively the visualization. Then the script checks access rights and builds the HTTP GET requests to be sent. Finally, the script transfers these requests to the device editor, which actually performs the sending.

2.1 Execution of Write Commands

Important: Transferring the HTTP GET requests to the device editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device!

The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

2.2 Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no HTTP GET requests will be created. Instead, the current value of the datapoint which is the value last sent, will be reported back.

The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
Device Editor

Process Interface Online Weather Request

Version 2

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Configuration....5
2. Datapoint Folders....6
   2.1. Datapoint Folder 'Current Weather'....6
   2.2. Datapoint Folder 'Location'....6
   2.3. Datapoint Folder 'Forcast'....6
3. Running the "Online Weather Request" Process Interface....7
   3.1 Access Control....7
   3.2 Execution of Write Commands....7
   3.3 Execution of Read Commands....7

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "Online Weather Request" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "Online Weather Request" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "Online Weather Request" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

The process interface example uses the restful API of the 'Free Plan' of https://openweathermap.org.

Functionality

The “Online Weather Request” process interface shows an example of how a data record formatted in JSON can be queried from a server and then evaluated by the script interface.

The script shows the queries and the evaluation of responses on the example of the API of OpenWeatherMap (https://openweathermap.org), a JSON/XML restful API, which is freely available at the time of creation of this process interface.

The prerequisite for using this process interface is an account at OpenWeatherMap, that allows the data request via the freely usable API (with up to 60 API calls per minute). Basis of the script is the API described in the documentation of the weather service (https://openweathermap.org/current and https://openweathermap.org/forecast5).

Furthermore, the software library "json.lua" is used, which is available for free use under the MIT license (https://github.com/rxi/json.lua).

The user can use this process interface to query current weather data and the forecast for up to 4 following days. The process interface provides several folders for this purpose: A configuration folder to configure the interface and several data point folders to store the received weather data.

The configuration folder contains the URL’s and an automatic query interval. By changing the value of the binary datapoint "Manual update" (for example by a switch control in the visualization), the update of the weather data can be carried out independently of the automatic interval.

1. Configuration

In order work properly, the URL's of the weather API must be entered in the configuration folder. These URL's contain the API key that is assigned when creating the OpenWeatherMap account. This API key is visible on the website of the account.

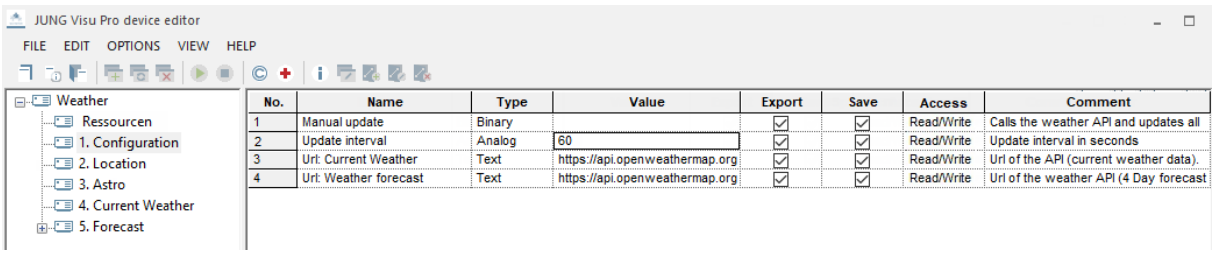
The URL's have the following structure:

https://api.openweathermap.org/data/2.5/<TYPE>?
zip=<ZIP>&units=metric&lang=en&appid=<API-KEY>

TYPE:	The type changes depending on whether the current weather or the forecast is retrieved. The script supports the following types: • For current weather: weather • For forecast: forecast
API-KEY:	Key assigned when the account is created.
ZIP:	Postcode of the city for which the weather data should be requested. Also contains the country code. Example: 58579, DE

The configuration folder can be selected by clicking in the directory tree. A table appears in the corresponding window on the right. Enter the URL's in the row "Url: Current weather" and "Url: Weather forecast".

In addition, an update interval of at least 60 seconds can be set. If this value remains empty, it is initialized with 3600 seconds by default.



After the process interface has been started, a request is sent to the entered URL. If you have entered the correct URL's, the process interface receives the JSON formatted weather data as response and evaluates it.

2. Datapoint Folders

After the process interface creation the datapoint tree provides the following folders: 'Current weather', 'Location' and the 'Forecast' for one or more days.

2.1. Datapoint Folder 'Current Weather'

Contains the weather data of the current day.

2.2. Datapoint Folder 'Location'

Contains additional information about the current location. This information (e.g. geographical longitude / latitude, time zone) will be provided by the weather service.

2.3. Datapoint Folder 'Forcast'

The datapoint folder 'Forecast', subfolder 'Day forecast', contains the weather data of the next days. When the process connection is created, only one day folder is already created by default. The number of days required to evaluate all information from the API response is 4 full days.

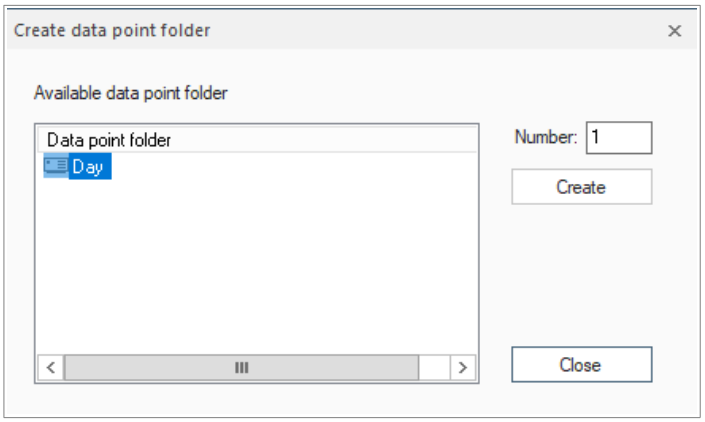
Creation of a new day folder:

- select the folder 'Day forecast' in the tree structure,
- open the context menu via right mouse click,
- select the command ***Create datapoint folder*****,**

alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create datapoint folder*** opens. Select the datapoint type ***Day*** and click the button ***Create***.

The names of all datapoints and datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.



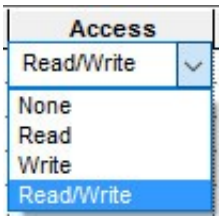
Each day contains folders for detailed weather data at 3 hour intervals and a general forecast.

3. Running the "Online Weather Request" Process Interface

To start the "Online Weather Request" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization, the weather data will be queried automatically from the weather service. The resulting data will be evaluated and displayed in the related datapoint folders.

3.1 Access Control



The script evaluates access rights. The predefined access rights should not be changed. Write access should only be assigned to the variables in the configuration folder.

3.2 Execution of Write Commands

Important: Transferring the HTTP requests to the device editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the weather service!

The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

3.3 Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no requests will be created. Instead, the current value of the datapoint which is the value last received, will be reported back.

The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
Device Editor

Process Interface Oracle Fidelio

Version 1.0

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Creation of the Data Point Folders....4
   1.1 Data Point Folder Room....5
   1.2 Data Point Folder Common....6
   1.3 Access Control....6
   1.4 Data Point Folder Resources....6
2. Running the Process Interface....7
   2.1 Execution of Write Commands....7
   2.2 Execution of Read Commands....7

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the **Oracle Fidelio Interface** described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The **Oracle Fidelio Interface** is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these term, you are not allowed to use the Oracle Fidelio Interface.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface ***Oracle Fidelio*** provides data points for Oracle Fidelio hotel solutions, which receive states via UDP. It is possible to create multiple rooms in an interface. For each room the interface will receive and send messages, for example to change the set temperature in the room.

1. Creation of the Data Point Folders

After creating the process interface, a default data point folder ***Floor*** is shown in the data tree.

A ***Floor*** folder can contain any number of ***Room*** folders. To create a new floor, click with the right key of your mouse on the process interface and select „Create a new data point folder". The same can be done to create new rooms, by right clicking on a floor.

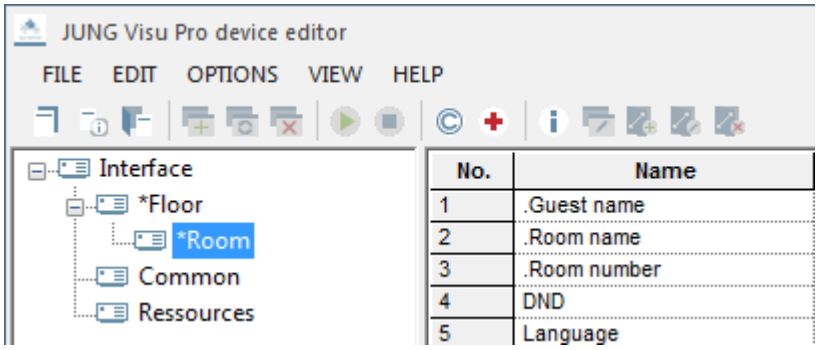
Important:
Every room in the process interface needs to contain a valid **room number**. This number has to be **unique**. If no room number is entered or if a number is used twice the interface will throw a configuration error!

Alongside the floors exists another data point folder named ***Common***. This folder contains variables, which are necessary for the interface to work.

The ***Resources*** folder contains the communication port which is monitored to receive new messages.

The data point folders ***Floor*** and ***Room*** as well as the ***Common*** folder can be renamed. The name of the folders do not matter for the evaluation of the script.

Every folder ***Room*** contains data points, which monitor the usage of the door mechanism.



1.1 Data Point Folder Room

Following data points are available in a room:

Room name:
Name of the room.

Room number:
Number of the room. Has to be unique across all rooms!

Guest name:
Name of the guest in the room.
Data point of the Oracle Fidelio gateway.

Language:
The displayed language in the room.
Data point of the Oracle Fidelio gateway.

DND:
Short form of 'Do Not Disturb'.
Data point of the Oracle Fidelio gateway.
Will be 1, if the guest does not want to be disturbed.

MUR:
Short form of 'Make Up Room'.
Data point of the Oracle Fidelio gateway.

Operation Mode Comfort-Eco:
Shows the active operation mode.
Forwarded data point of the Oracle Fidelio gateway.

Operation Mode Comfort-Eco Bin:
Shows the active operation mode as a binary value.
Forwarded data point of the Oracle Fidelio gateway.

Set temperature:
The desired set temperature of the guest.
Data point of the Oracle Fidelio gateway.

SOS:
Will be set to 1 in case of emergency.
Data point of the Oracle Fidelio gateway.

1.2 Data Point Folder Common

The data point folder ***Common*** contains following data points:

Configuration error:
Will be set to 1, if an error in the configuration occurs.

IP address:
The IP address of the JUNG Gateway.
Has to be set, if **Use IP address as UDP filter** is used.

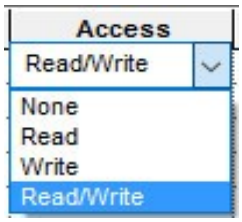
Runtime error:
Will be set to 1, if an error occurs at runtime.

Sending port:
Port of the JUNG Gateway-Application to receive commands from the interface.

Use IP address as UDP filter:
If enabled, the interface will only evaluate UDP messages which are send from the given IP address.
If left empty, it will be set to 1 on initialization.

Important:
The interface will only work, if **all room numbers** are set.
If **Use IP address as UDP filter** is used, the interface will not start as long as no IP address is entered.
In this cases the data point **Configuration error** will be set to **1**.

1.3 Access Control



The script evaluates access rights. Datagrams will only be sent when access rights **write** or **read/write** are specified for the data point. You should not change the access rights in this script.

1.4 Data Point Folder Resources

The folder Resources contains the address configuration for the communication port. The telegrams will be received on this port (default: 5702).

2. Running the Process Interface

The process interface can be started with **Menu – Options – Start process interface** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

If all parameters are set correctly, the interface will wait for incoming UDP commands and evaluate them. At the configured time all values will be reset daily. Before that they will be written into the protocol, if enabled.

2.1 Execution of Write Commands

Important: Transferring the datagrams to the devices editor is the last operation of the script regarding sending. Afterward execution of the command is reported as successful to the process model. There is no check whether the data could be sent and have been received by the device!

With access rights **write** or **read/write** the new value sent will be reported back to the process model, otherwise not.

2.2 Execution of Read Commands

When a read command is invoked by the process model respectively the user, the actual value of the data point, which is the last send value, will be reported back.

The actual value will be reported back only if the access rights **read** or **read/write** have been set.

JUNG Visu Pro
Device Editor

Process Interface PI-Controller

Version 5.0

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Creation of the Data Point Folder PI-Controller....5
   1.1 Data Point Folder PI-Controller....5
      1.1.2 Access Control....9
   1.2 Data Point Folder Resources....10
2. Running the Process Interface....10
   2.1 Execution of Write Commands....10
   2.2 Execution of Read Commands....10
3. Receiving Values from the Smart Control IP....11
   3.1 Send via Timer Switch....11
   3.1 Sending the Master Button....12

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the **PI-Controller** software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The **PI-Controller** software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these term, you are not allowed to use the PI-Controller software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface **PI-Controller** provides you with one or more dynamic PI controller with heating function for room temperature control.

The controller manipulates the valve position in such a way, that the set room temperature, also by disturbing influences, is maintained as good as possible.

To ensure that the controller compares the set temperature with the current room temperature, he gets and calculates the values of **temperature difference**, the **proportional band** and the **reset time** a position value for the heating valve between 0 and 100 %.

The implemented controller is a dynamic PI-Controller and consists of a proportional part (P portion) and an integral part (I portion).

The P portion determines the manipulated variable proportionally to the control deviation. So the position value is a multiplication of the temperature difference and a gain factor.

> **Small proportional band**: Manage the controller deviation quickly (small temperature differences are enough to raise the manipulated variable quickly to a high value). If the chosen proportional band is to small there is a danger of overshooting the desired set temperature.
>
> **Large proportional band**: Manage the controller deviation slowly. There is nearly no danger of overshooting the desired set temperature.

The I portion takes the change over time into account and leads to an integral approach of the current temperature to the set temperature. He sums up the control deviation over time and multiplies this summed up control deviation with the factor, which is calculated from the proportional portion and the reset time.

Anti-Windup

The I portion will not further summed up, if the manipulated variable is really close to overshoot 100% or to fall below 0%.

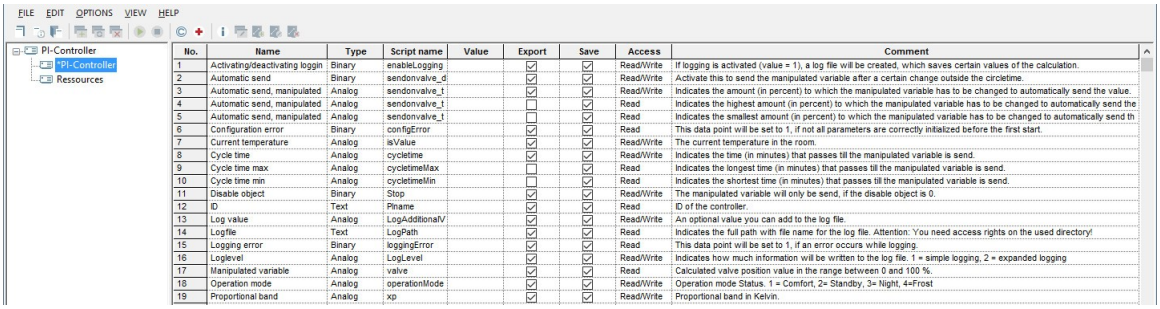
1. Creation of the Data Point Folder PI-Controller

After creating the process interface, a default data point folder **PI-Controller** is shown in the tree.

You can create more than one **PI-Controller** folder in the same interface by right clicking on the parent node of the folder and choosing to create a new data point folder in the appearing context menu.

The data point folder **PI-Controller** can be renamed. For example this can be used to show in which room the selected PI controller is located. The name of the folder does not matter for the evaluation of the script.

The folder **PI-Controller** contains all necessary data points for one PI-Controller.



No.	Name	Type	Script name	Value	Export	Save	Access	Comment
1	Activating/deactivating loggin	Binary	enableLogging		☑	☑	Read/Write	If logging is activated (value = 1), a log file will be created, which saves certain values of the calculation.
2	Automatic send	Binary	sendonvalve_d		☑	☑	Read/Write	Activate this to send the manipulated variable after a certain change outside the circletime.
3	Automatic send, manipulated	Analog	sendonvalve_t		☑	☑	Read/Write	Indicates the amount (in percent) to which the manipulated variable has to be changed to automatically send the value.
4	Automatic send, manipulated	Analog	sendonvalve_t		☐	☑	Read	Indicates the highest amount (in percent) to which the manipulated variable has to be changed to automatically send the
5	Automatic send, manipulated	Analog	sendonvalve_t		☐	☑	Read	Indicates the smallest amount (in percent) to which the manipulated variable has to be changed to automatically send th
6	Configuration error	Binary	configError		☑	☑	Read	This data point will be set to 1, if not all parameters are correctly initialized before the first start.
7	Current temperature	Analog	isValue		☑	☑	Read/Write	The current temperature in the room.
8	Cycle time	Analog	cycletime		☑	☑	Read/Write	Indicates the time (in minutes) that passes till the manipulated variable is send.
9	Cycle time max	Analog	cycletimeMax		☐	☑	Read	Indicates the longest time (in minutes) that passes till the manipulated variable is send.
10	Cycle time min	Analog	cycletimeMin		☐	☑	Read	Indicates the shortest time (in minutes) that passes till the manipulated variable is send.
11	Disable object	Binary	Stop		☑	☑	Read/Write	The manipulated variable will only be send, if the disable object is 0.
12	ID	Text	PIname		☑	☑	Read	ID of the controller.
13	Log value	Analog	LogAdditionalV		☑	☑	Read/Write	An optional value you can add to the log file.
14	Logfile	Text	LogPath		☑	☑	Read	Indicates the full path with file name for the log file. Attention: You need access rights on the used directory!
15	Logging error	Binary	loggingError		☑	☑	Read	This data point will be set to 1, if an error occurs while logging.
16	Loglevel	Analog	LogLevel		☑	☑	Read/Write	Indicates how much information will be written to the log file. 1 = simple logging, 2 = expanded logging
17	Manipulated variable	Analog	value		☑	☑	Read	Calculated valve position value in the range between 0 and 100 %.
18	Operation mode	Analog	operationMode		☑	☑	Read/Write	Operation mode Status. 1 = Comfort, 2= Standby, 3= Night, 4=Frost
19	Proportional band	Analog	xp		☑	☑	Read/Write	Proportional band in Kelvin.

1.1 Data Point Folder PI-Controller

Every controller in the interface needs several input values and parameters to calculate the valve value. The data point folder **PI-Controller** uses 39 data points.

Thirtytwo of these data points can manage input of the user and have to be set before starting the script:

Input Values For The PI-Controller

Current temperature:
The current temperature in the room. For example 18(°C).

Set temperature:
The temperature the controller tries to reach and maintain. For example 21(°C).
Accepted value range if not setup manually: 0 – 50 °C.

Set temperature comfort:
The temperature for the operation mode "comfort".
If this value does not get set by the user, it will be set to 21 automatically.
Accepted value range if not setup manually: refer set temperature

Set temperature standby:

The temperature for the operation mode "standby".
If this value does not get set by the user, it will be set to 19 automatically.
Accepted value range if not setup manually: refer set temperature

Set temperature night:

The temperature for the operation mode "night".
If this value does not get set by the user, it will be set to 17 automatically.
Accepted value range if not setup manually: refer set temperature

Set temperature frost:

The temperature for the operation mode "frost".
If this value does not get set by the user, it will be set to 7 automatically.
Accepted value range if not setup manually: refer set temperature

Operation mode:

The set temperature will be set to the respective value of the operation mode.
Value 1: Comfort
Value 2: Standby
Value 3: Night
Value 4: Frost
Accepted value range: 1 to 4

Reset time:

Indicates the time (in minutes) needed to double the valve position value, if the temperature difference stays the same. For example: An underfloor heating system has a reset time of 240 minutes.
If this value does not get set by the user, it will be set to 150 automatically.
Accepted value range if not setup manually: 0 – 720 minutes.

Proportional band:

Indicates the temperature range in which the controller controls the valve position value. This means: It indicates the change of the current temperature that is needed to open the valve from 0% to 100%. Is the current temperature not in between this temperature range, the valve position will not be changed (0% respectivly 100%).

For example: An underfloor heating system uses a proportional band of 5(K) and the user defines the set temperature to 20°C. In this case the valve would be completely open, if the current temperature is under 15°C and completely shut, if it overshoots the set temperature.
If this value does not get set by the user, it will be set to 5 automatically.
Accepted value range if not setup manually: 0 – 20 K.

Examples for **reset time** and **proportional band**:

warm water heating system:	5K / 150 min
underfloor heating system:	5K / 240 min
electric heating system:	4K / 100 min
fan coil:	4K / 90 min
split-unit:	4K / 90 min

Disable object:

The manipulated variable will only be send, if the **disable object** is deactivated (value = 0). This does not affect the calculation of the manipulated variable.
If this value does not get set by the user, it will be set to 0 automatically.

Temperature adjustment:

Adjustment value for the current temperature.
If this value does not get set by the user, it will be set to 0 automatically.
Accepted value range if not setup manually: -20 to 20 °C.

Cycle time:

Indicates the time (in minutes) that passes till the manipulated variable is send.
If this value does not get set by the user, it will be set to 2 automatically.
Accepted value range if not setup manually: 1 – 40 minutes.

Automatic send:

If **automatic send** is activated (value = 1), the manipulated variable will always be send, if it changes by a certain amount (in percent) during the calculations.
If this value does not get set by the user, it will be set to 1 automatically.

Automatic send, manipulated variable change:

Indicates the amount (in percent) to which the manipulated variable has to be changed to automatically send the value outside of the cycle time.
You only need to set this value, if **automatic send** is active.
If this value does not get set by the user, it will be set to 5 automatically.
Accepted value range if not setup manually: 0 – 100 percent.

Activating/deactivating logging:

If logging is activated (value = 1), a log file will be created, which saves certain values of the calculation.
If this value does not get set by the user, it will be set to 0 automatically.

Logfile:

Indicates the full path with file name for the log file.
Attention: You need access rights on the used directory!
Example: C:\Log\Logfile.csv

Loglevel:

Indicates how much information will be written to the log file.
If this value does not get set by the user, it will be set to 1 automatically.

At 1: simple logging

- time
- manipulated variable
- set temperature
- adjusted current temperature
- additional log value

At 2: expanded logging

There will additional values be written.
For example:

- temperature adjustment
- current temperature
- old and new temperature differences
- old and new P portion
- old and new integrated areas
- the I portion of the current pulse interval
- summed up I portion
- if the manipulated variable has been send or not

Log value:

An optional value you can add to the log file. This value does not affect the calculation of the manipulated variable.
There you could add the current outside temperature for example.

UDP Current temperature Enable:

Set this data point to 1, if the current temperature should be received over UDP. If this is active, you can not use the process model to change the value of the current temperature.

UDP: IP address of the sending device:

Indicates the IP address of the device, which sends the commands over UDP. Current temperature values will only be handled, if **UDP Current temperature Enable** is activated.
For example: 192.168.0.10

Output Values of the PI-Controller:

Manipulated variable:
Calculated valve position value in the range between 0 and 100 %.

ID:
Can be used to differentiate multiple controllers in the same interface.
If this value does not get set by the user in the interface, the controller gets the default ID **Controller**.

Configuration error:
This data point will be set to 1, if not all parameters are correctly initialized before the first start.

Runtime error:
This datapoint will be set to 1, if an error occurs during the runtime.

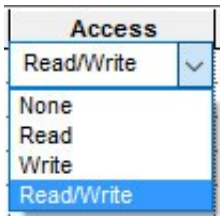
Logging error:
This datapoint will be set to 1, if an error occurs while logging.

Set temperature status:
This data point shows the status of the set temperature.

SCIP master button:
Will be set to 1, if the master button of the Smart Control ip is pressed and is set to 0, if it is released. To use this datapoint you have to enter the ip address of the Smart Control IP into **UDP: IP address of the sending device**.

Important:
The controller only works properly, if **current temperature**, **set temperature**, **reset time** and **proportional band** are set.
If this is not the case, there will be a **configuration error.**

1.1.2 Access Control



The script evaluates access rights. Data grams will only be sent when access rights **write** or **read/write** are specified for the data point. You should not change the access rights in this script.

1.2 Data Point Folder Resources

The folder Resources contains the address configuration for the communication-port. The telegrams will be received on this port (default: 42000).

The value of the current temperature is send as a ASCII text. The incoming telegrams will be matched to a string pattern to single out the ones containing values for the current temperature. A valid value for the current temperature begins with a (+/-) number '.' or ',' and another number.

Valid temperature values are for example:

19.0
-1.7
19,0
+17,8

2. Running the Process Interface

The process interface can be started with **Menu – Options – Start process interface** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

If all parameters are set correctly, the controller will calculate a temperature difference and the manipulated variable every minute. as long as the disable object is not active, the manipulated variable will be send after the set cycle time or after a certain change to it's previous value.

2.1 Execution of Write Commands

Important: Transferring the data grams to the devices editor is the last operation of the script regarding sending. Afterward execution of the command is reported as successful to the process model. There is no check whether the data could be sent and have been received by the device!

With access rights **write** or **read/write** the new value sent will be reported back to the process model, otherwise not.

2.2 Execution of Read Commands

When a read command is invoked by the process model respectively the user, the actual value of the data point, which is the last send value, will be reported back.

The actual value will be reported back only if the access rights **read** or **read/write** have been set.

3. Receiving Values from the Smart Control IP

It is possible to send values over UDP to the PI-Controller with the "Master button" and the timer switch of the Smart Control IP, as long as you entered a valid IP address.

3.1 Send via Timer Switch

Following options have to be entered to send values over the timer switch:

Type: Choose "Binary", if you want to send values to a binary datapoint. Otherwise choose a fitting value range.

Display Minimal/ Maximal: Enter the smallest and highest value you can send to the specified datapoint. **These values should be the same as the values you entered in the PI-Controller!**
The minimal and maximal settings for the operation mode must be 1 and 4.

Command Address: The command for the specified datapoint. There are the following commands:

Command Address	Datapoint	Type
pi_setvalue	Set temperature	Analogue
pi_integral	Reset time	Analogue
pi_proportional	Proportional band	Analogue
pi_lock	Disable object	Binary
pi_tempadjust	Temperature adjustment	Analogue
pi_cycletime	Cycle time	Analogue
pi_autosend	Automatic send	Binary
pi_autosendchangelimit	Automatic send, manipulated variable change	Analogue
pi_opcomfort	Set temperature comfort	Analogue
pi_opstandby	Set temperature standby	Analogue
pi_opnight	Set temperature night	Analogue
pi_opfrost	Set temperature frost	Analogue
pi_opmode	Operation mode	Analogue

Unit and Function type: These options should be left empty, respectively set to "No astro".

3.1 Sending the Master Button

To see, if the master button of the Smart Control Ip is pressed or released, you have to enter **pi_masterbutton** into "Command sequence on press" and "Command sequence on release". You find these configurations by entering the ip of your Smart Control IP into a web browser.

JUNG Visu Pro
Device Editor

Process Interface Protel

Version 1.0

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Creation of the Data Point Folders....4
   1.1 Data Point Folder Room....5
   1.2 Data Point Folder Common....6
   1.3 Access Control....6
   1.4 Data Point Folder Resources....6
2. Running the Process Interface....7
   2.1 Execution of Write Commands....7
   2.2 Execution of Read Commands....7

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the **Protel Interface** described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The **Protel Interface** is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these term, you are not allowed to use the Protel Interface.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface ***Protel*** provides data points for Protel hotel solutions, which receive states via UDP. It is possible to create multiple rooms in an interface. For each room the interface will receive and send messages, for example to change the set temperature in the room.

1. Creation of the Data Point Folders

After creating the process interface, a default data point folder ***Floor*** is shown in the data tree.

A ***Floor*** folder can contain any number of ***Room*** folders. To create a new floor, click with the right key of your mouse on the process interface and select „Create a new data point folder". The same can be done to create new rooms, by right clicking on a floor.

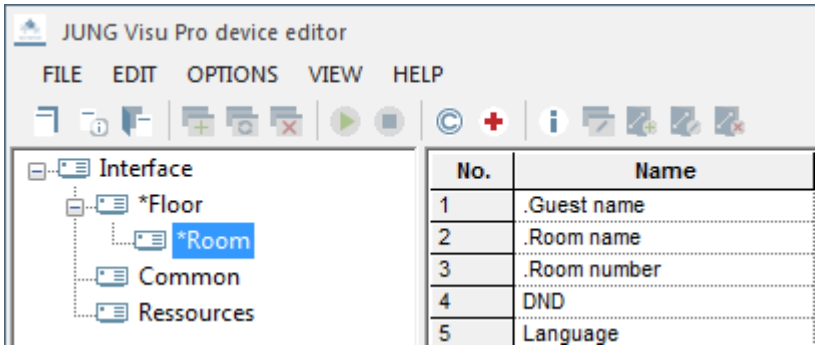
Important:
Every room in the process interface needs to contain a valid **room number**. This number has to be **unique**. If no room number is entered or if a number is used twice the interface will throw a configuration error!

Alongside the floors exists another data point folder named ***Common***. This folder contains variables, which are necessary for the interface to work.

The ***Resources*** folder contains the communication port which is monitored to receive new messages.

The data point folders ***Floor*** and ***Room*** as well as the ***Common*** folder can be renamed. The name of the folders do not matter for the evaluation of the script.

Every folder ***Room*** contains data points, which monitor the usage of the door mechanism.



1.1 Data Point Folder Room

Following data points are available in a room:

Room name:
Name of the room.

Room number:
Number of the room. Has to be unique across all rooms!

Guest name:
Name of the guest in the room.
Data point of the Protel gateway.

Language:
The displayed language in the room.
Data point of the Protel gateway.

DND:
Short form of 'Do Not Disturb'.
Data point of the Protel gateway.
Will be 1, if the guest does not want to be disturbed.

MUR:
Short form of 'Make Up Room'.
Data point of the Protel gateway.

Operation Mode Comfort-Eco:
Shows the active operation mode.
Forwarded data point of the Protel gateway.

Operation Mode Comfort-Eco Bin:
Shows the active operation mode as a binary value.
Forwarded data point of the Protel gateway.

Set temperature:
The desired set temperature of the guest.
Data point of the Protel gateway.

SOS:
Will be set to 1 in case of emergency.
Data point of the Protel gateway.

1.2 Data Point Folder Common

The data point folder ***Common*** contains following data points:

Configuration error:
Will be set to 1, if an error in the configuration occurs.

IP address:
The IP address of the JUNG Gateway.
Has to be set, if **Use IP address as UDP filter** is used.

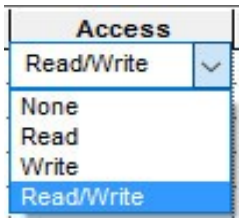
Runtime error:
Will be set to 1, if an error occurs at runtime.

Sending port:
Port of the JUNG Gateway-Application to receive commands from the interface.

Use IP address as UDP filter:
If enabled, the interface will only evaluate UDP messages which are send from the given IP address.
If left empty, it will be set to 1 on initialization.

Important:
The interface will only work, if **all room numbers** are set.
If **Use IP address as UDP filter** is used, the interface will not start as long as no IP address is entered.
In this cases the data point **Configuration error** will be set to **1**.

1.3 Access Control



The script evaluates access rights. Datagrams will only be sent when access rights **write** or **read/write** are specified for the data point. You should not change the access rights in this script.

1.4 Data Point Folder Resources

The folder Resources contains the address configuration for the communication port. The telegrams will be received on this port (default: 5702).

2. Running the Process Interface

The process interface can be started with **Menu – Options – Start process interface** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

If all parameters are set correctly, the interface will wait for incoming UDP commands and evaluate them. At the configured time all values will be reset daily. Before that they will be written into the protocol, if enabled.

2.1 Execution of Write Commands

Important: Transferring the datagrams to the devices editor is the last operation of the script regarding sending. Afterward execution of the command is reported as successful to the process model. There is no check whether the data could be sent and have been received by the device!

With access rights **write** or **read/write** the new value sent will be reported back to the process model, otherwise not.

2.2 Execution of Read Commands

When a read command is invoked by the process model respectively the user, the actual value of the data point, which is the last send value, will be reported back.

The actual value will be reported back only if the access rights **read** or **read/write** have been set.

JUNG Visu Pro
Device Editor

Process Interface „TCP Commands“

Version 1

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Connection to the Server....4
   2.1. Device Folder....5
   2.2. Creation of Datapoint Folders....5
      2.2.1. Datapoint Folder *Switch on/off*....6
      2.2.2. Datapoint Folder *Analog Value*....6
      2.2.3. Datapoint Folder *Date/Time*....7
      2.2.4. Using Numerical Values in Command Formats....7
      2.2.5. Access Control....8
3. Running the "TCP commands" Process Interface....8
   3.1 Execution of Write Commands....8
   3.2 Execution of Read Commands....8

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "TCP Commands" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "TCP Commands" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "TCP commands" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface "TCP commands" allows to send TCP data packets over the network to a TCP server.

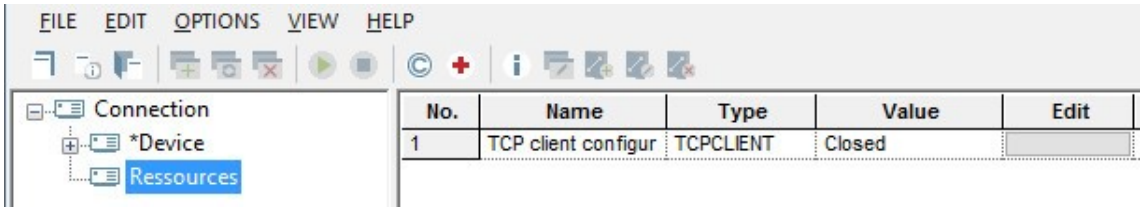
The contents and meaning of the TCP data packets depend on the specific device, but the user can specify the contents of the data packets in order to implement binary switch, analog set and date/time set commands and corresponding data points.
These datapoints reside in the datapoint folders.

When using the "TCP commands" software, it may be necessary to configure firewalls, router and other devices to enable the sending of TCP telegrams over the network by the software. These measures are the user's responsibility and are not covered here.

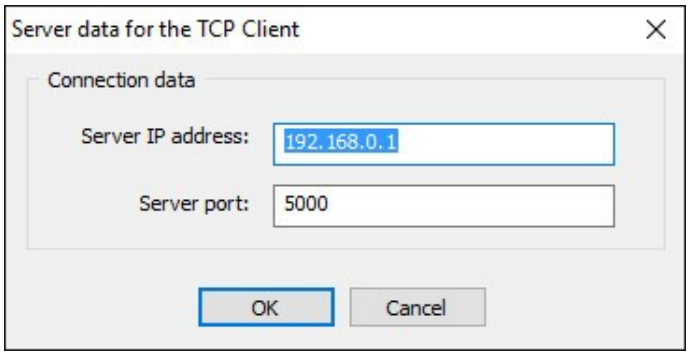
1. Connection to the Server

To communicate with a TCP server, the process interface needs the IP address and port of the server.

Click on the resource tree item to find the properties of the TCP communication port in the resources data table.



Click the button in the row ***TCP Client configuration***, column ***Edit*** to see or change the port number used.



The port number should not be used by another application and its status should switch from 'closed' to 'open' when the process interface starts. It will remain open until the process interface is stopped. The status is visible in the column ***value*** in the row of the communication port.

Note: The name of the resource folder cannot be changed.

2.1. Device Folder

The device folder is created including already the subfolders ***Analog value***, ***Date/Time***, ***Switch on/off*** datapoint folders for possible commands and corresponding datapoints of the respective type.

2.2. Creation of Datapoint Folders

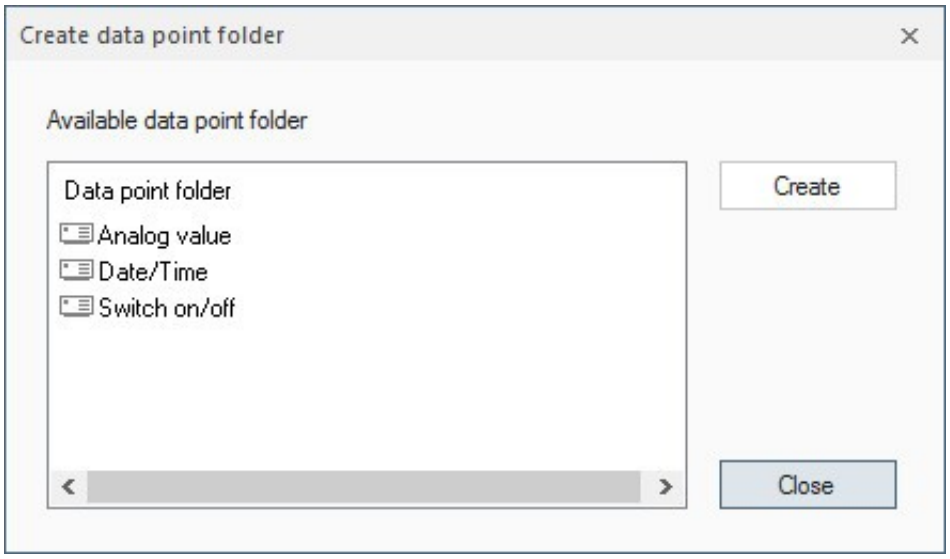
Each datapoint of a device is represented with a datapoint folder inside of the device folder. When the data point folder is selected with a left click, a table of the datapoint properties is displayed.

In order to define more data points and in addition to the default datapoint folders, more datapoint folders can be created as subitems to the device folder.

To create a new datapoint folder, select the device folder in the structure tree, then click the right mouse button to open its context menu. In the context menu select the command ***Create datapoint folder***. Alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create data point folder*** opens. Select the desired datapoint type and click the button ***Create***.

The datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.



Note: If the process interface is started, no new datapoints can be added.

2.2.1. Datapoint Folder *Switch on/off*

This folder contains three data points:

- **Switch**

  This data point is of the type 'binary' and is exported per default to the process model. Exporting means that it will be visible in the process model as a regular binary datapoint. It will be visible in the visualization editor if the process model exports the data point as well.

  When the datapoint is switched **OFF**, the script will send the text for entry ***switch command off***.

  Switching **ON** is similar: the text for entry ***switch command on*** will be sent to the device.

- ***Important:*** The ***save*** flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn off the device.

  If the **export** flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default the export flag is not set.

2.2.2. Datapoint Folder *Analog Value*

This folder contains two data points:

- **Analog Value**

  This datapoint is representing an analog value. It is exported per default to the process model where it is a regular analog data point.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the ***Command text*** entry.

- **Command Text**

  The ***Command text*** entry defines the command string to be sent to the device when the value of the analog datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the *%d* placeholder. Before sending, the script will replace *%d* with the actual value.

- ***Important:*** The ***save*** flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

  If the **export** flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default, the export flag is not set.

2.2.3. Datapoint Folder *Date/Time*

This folder contains two data points:

- **Date/Time**

  The type of this data point is *Date/Time*. Per default, it will be exported to the process model.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the ***Command text*** entry.

- **Command Text**

  The ***Command text*** entry defines the command string to be sent to the device when the value of date/time datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the *%s* placeholder. Before sending, the script will replace *%s* with the actual value.

- ***Important:*** The ***save*** flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be send.

  If the **export** flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default, the export flag is not set.

2.2.4. Using Numerical Values in Command Formats

In command formats it is possible to enter characters as numerical values. A substring (called "escape sequence") in format ***\ddd***, where ***ddd*** must be a 1 – 3 digit value between 1 and 255, will be transmitted as a byte with this value.

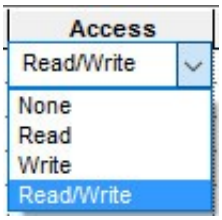
Examples:

\10 encodes a linefeed
\64 encodes the at sign '@'

Note that a byte with the value 0 cannot be used.
If the char '\' is to be inserted literally, it must be duplicated.

Format ***\\001*** will send ***\001*** whereas ***\001*** will send the byte with value 1.

2.2.5. Access Control



The script evaluates access rights. TCP datagrams will only be sent when access rights ***Write*** or ***Read/Write*** are specified for the datapoint.

3. Running the "TCP commands" Process Interface

To start the "TCP commands" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device succeeds, the script will recognize changes carried out on the datapoints in the process model respectively the visualization. Then the script checks access rights and builds the strings to be sent. Finally, the script transfers these telegrams to the devices editor, which actually performs the sending.

3.1 Execution of Write Commands

Important: Transferring the TCP datagrams to the devices editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device!

The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

3.2 Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no TCP telegram will be created. Instead, the current value of the data point which is the value last sent, will be reported back.

The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
Device Editor

Process Interface „UDP Commands“

Version 1

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Specification of the Local Communication Port....4
2. Datapoints of an IP-Capable Device....5
   2.1. Device Folder....5
   2.2. Creation of Datapoint Folders....6
      2.2.1. Datapoint Folder *Switch on/off*....6
      2.2.2. Datapoint Folder *Analog Value*....7
      2.2.3. Datapoint Folder *Date/Time*....8
      2.2.4. Using Numerical Values in Command Formats....8
      2.2.5. Access Control....9
3. Running the "UDP Commands" Process Interface....9
   3.1 Execution of Write Commands....9
   3.2 Execution of Read Commands....9

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "UDP Commands" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "UDP Commands" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "UDP commands" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface "UDP commands" allows to send UDP data packets over the Internet to IP-capable devices, with each an own IP address. UDP ("User Datagram Protocol") is fundamental to the IP network and does not presume a certain structure of the UDP data packets.

The contents and meaning of the UDP data packets depend on the specific device, but the user can specify the contents of the data packets in order to implement binary switch, analog set and date/time set commands and corresponding data points.
These datapoints reside in the datapoint folders.

Moreover, sending UDP data packets requires a local communication port. This port and similar information resides in the resources folder.

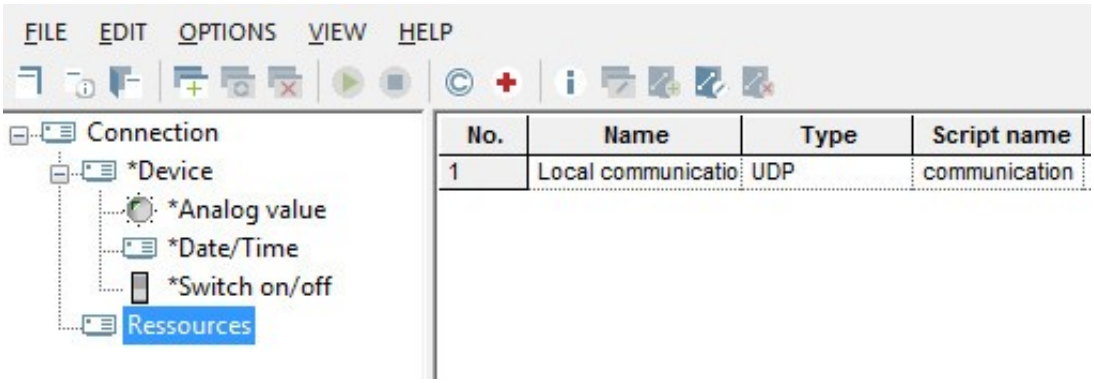
When using the "UDP commands" software, it may be necessary to configure firewalls, router and other devices to enable the sending of UDP telegrams over the network by the software. These measures are the user's responsibility and are not covered here.

1. Specification of the Local Communication Port

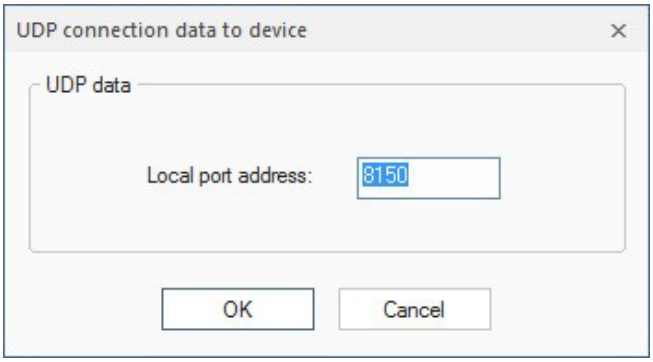
As part of the Internet data protocol, most notably the Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP), a number is used to distinguish concurrent communications between the endpoints of an IP connection. The endpoint is called a port, the number is called the port number. With different ports, a computer can virtually at the same time connect to a web server, receive e-mails, and use or provide other services.

Some port numbers are reserved for special purposes, for example the port number 80 is used for the http protocol.

For the "UDP commands" software, a free port number must be found and assigned. It can be specified in the resources folder.
Click on the resource tree item to find the properties of the UDP communication port in the resources data table.



Click the button in the row ***communication port***, column ***Edit*** to see or change the port number used.



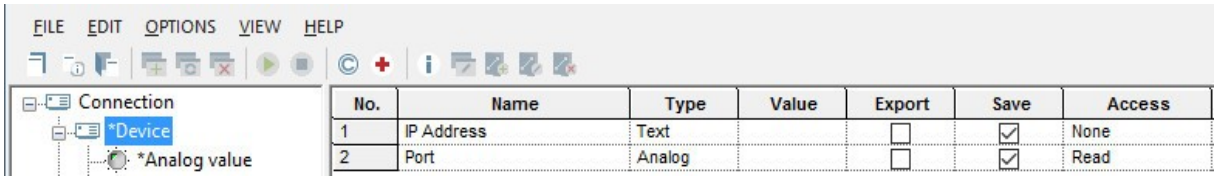
The port number should not be used by another application and its status should switch from 'closed' to 'open' when the process interface starts. It will remain open until the process interface is stopped. The status is visible in the column ***value*** in the row of the communication port.

The name of the resource folder cannot be changed.

2. Datapoints of an IP-Capable Device

After a process interface of the "UDP-Commands" type has been created, a new tree node with the name ***device*** is added. The node represents an IP-capable device. From hereafter we will call this node the "device node", though in the actual project it should be renamed to something more descriptive, for example "beamer", "remote control" or "sound system" or "sound system volume" depending on the device and application.
Note that different command/datapoint sets of one device or different devices may be described each in different datapoint folders.

The asterisk in front of the node name signals that more than one node of this kind may be created by right-clicking its tree parent node and opening the context menu.



2.1. Device Folder

Basic information regarding the device includes its IP address in text form, for example 192.168.0.110. If an IP name has been assigned, which resolves to this address, the name can be used instead of the address.

In addition to the address, a port number must be specified to properly define the recipient of the UDP data packet.

With the port specified in the resources folder (and implicitly the computer IP address) for the sender and both, IP address and port of the IP-capable device, as the receiver, both end points of the IP communication are defined.

The device folder is created including already the subfolders ***Analog value***, ***Date/Time***, ***Switch on/off*** datapoint folders for possible commands and corresponding datapoints of the respective type.

2.2. Creation of Datapoint Folders

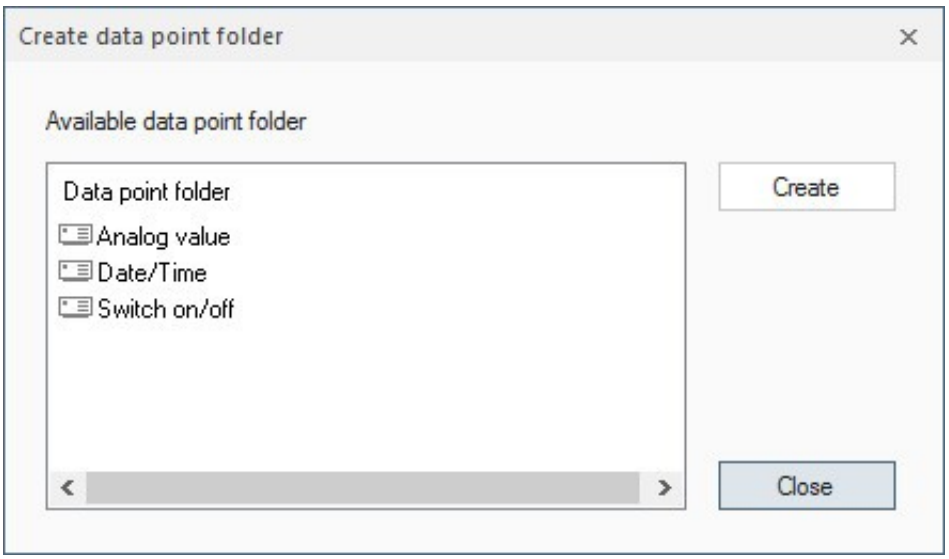
Each datapoint of a device is represented with a datapoint folder inside of the device folder. When the data point folder is selected with a left click, a table of the datapoint properties is displayed.

In order to define more data points and in addition to the default datapoint folders, more datapoint folders can be created as subitems to the device folder.

To create a new datapoint folder, select the device folder in the structure tree, then click the right mouse button to open its context menu. In the context menu select the command ***Create datapoint folder***. Alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create data point folder*** opens. Select the desired datapoint type and click the button ***Create***.

The datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.



2.2.1. Datapoint Folder *Switch on/off*

This folder contains three data points:

- **Switch**

  This data point is of the type 'binary' and is exported per default to the process model. Exporting means that it will be visible in the process model as a regular binary datapoint. It will be visible in the visualization editor if the process model exports the data point as well.

  When the datapoint is switched off, the script will send the text for entry 'switch command off'. Switching on is similar: the text for entry 'switch command on' will be sent to the device.

- **Switch OFF command**

  The value of this variable is the text which will be sent for the logical state 'OFF'.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn off the device.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default the export flag is not set.

- **Switch ON command**

  The value of this variable is the text which will be sent for the logical state 'ON'.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn on the device.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default the export flag is not set.

2.2.2. Datapoint Folder *Analog Value*

This folder contains two data points:

- **Analog Value**

  This datapoint is representing an analog value. It is exported per default to the process model where it is a regular analog data point.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the 'Command Text' – entry.

- **Command Text**

  The 'Command Text' entry defines the command string to be sent to the device when the value of the analog datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the %d placeholder. Before sending, the script will replace %d with the actual value.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default, the export flag is not set.

2.2.3. Datapoint Folder *Date/Time*

This folder contains two data points:

- **Date/Time**

  The type of this data point is 'Date/Time'. Per default, it will be exported to the process model.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the 'Command Text' – entry.

- **Command Text**

  The 'Command Text' – entry defines the command string to be sent to the device when the value of date/time datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the %s placeholder. Before sending, the script will replace %s with the actual value.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be send.

  If the export-flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default, the export flag is not set.

2.2.4. Using Numerical Values in Command Formats

In command formats it is possible to enter characters as numerical values. A substring (called "escape sequence") in format \ddd, where ddd must be a 1 – 3 digit value between 1 and 255, will be transmitted as a byte with this value.

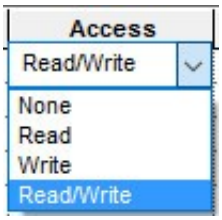
Examples:

\10 encodes a linefeed
\64 encodes the at sign '@'

Note that a byte with the value 0 cannot be used.
If the char '\' is to be inserted literally, it must be duplicated.

Format "\\001" will send "\001" whereas "\001" will send the byte with value 1.

2.2.5. Access Control



The script evaluates access rights. UDP datagrams will only be sent when access rights ***Write*** or ***Read/Write*** are specified for the datapoint.

3. Running the "UDP Commands" Process Interface

To start the "UDP commands" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device succeeds, the script will recognize changes carried out on the datapoints in the process model respectively the visualization. Then the script checks access rights and builds the strings to be sent. Finally, the script transfers these telegrams to the devices editor, which actually performs the sending.

3.1 Execution of Write Commands

Important: Transferring the UDP datagrams to the devices editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device!

The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

3.2 Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no UDP telegram will be created. Instead, the current value of the data point which is the value last sent, will be reported back.

The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
Device Editor

Process interface Byte to Bits Conversion

Version 1.0

Content:

Disclaimer 1
Contact person 2
Trademark 2
1. Functionality 3
2. Creating the data point folder Byte - Bits Conversion 4
Data point folder of the Byte – Bits Conversion 4
The input values of the Byte to Bits Conversion 4
The output values of the Bye to Bits Conversion 4
Setting the access rights 5
Data point folder resources 5
3. Carrying out the process interface 5
Execution of the write commands 5
Execution of the read commands 5
System news 5

Disclaimer

No warranty is given for the correctness and functionality of the process interface. Liability for damages or consequential damages resulting from the use of the process interface is excluded.

This also applies to program components that do not originate from the author of the RS flip-flop itself.

By using this process interface the user accepts the disclaimer!

Contact person

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89
mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All rights reserved

Trademark

EIB®, KNX® and ETS® are registered trademarks of KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the USA and other countries.
OPC® is a registered trademark of the OPC Foundation.
Sax Basic Engine is a registered trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trade names, company names and product names are trademarks or registered trademarks of their respective owners.

1. Functionality

The ***Byte - Bits Conversion*** process interface filters the status of every significant bit in a byte. Thus it can be determined whether the value of the bit is 0 or 1. The ***Byte - Bits Conversion*** thus converts a 1-byte variable into individual bits.

Several 1-byte variables can be connected to the process interface.

Every single variable of this process interface has the following objects:

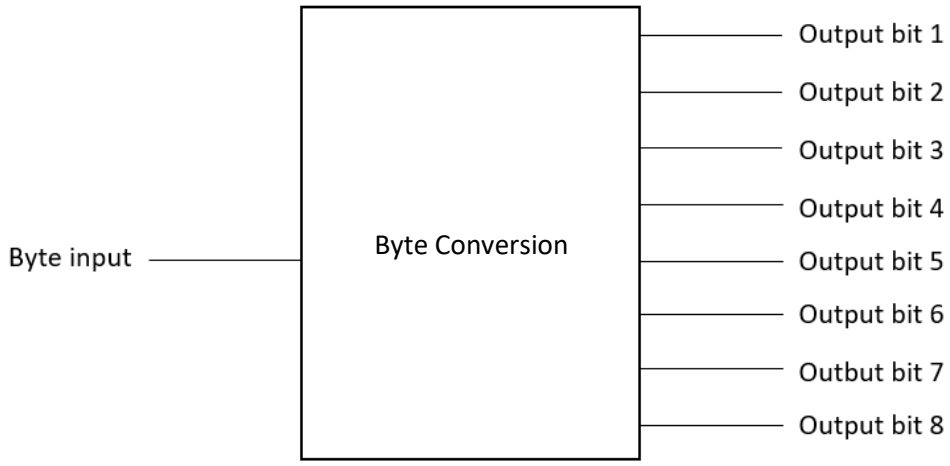


Figure 1

- **Byte input:** The value of the process variable is updated via this object.
- **Output bits 1 to 8:** These objects indicate the individual values of the corresponding bits.

2. Creating the data point folder Byte - Bits Conversion

After creating the process interface, a default data point folder ***Byte to Bits Converter*** is displayed.

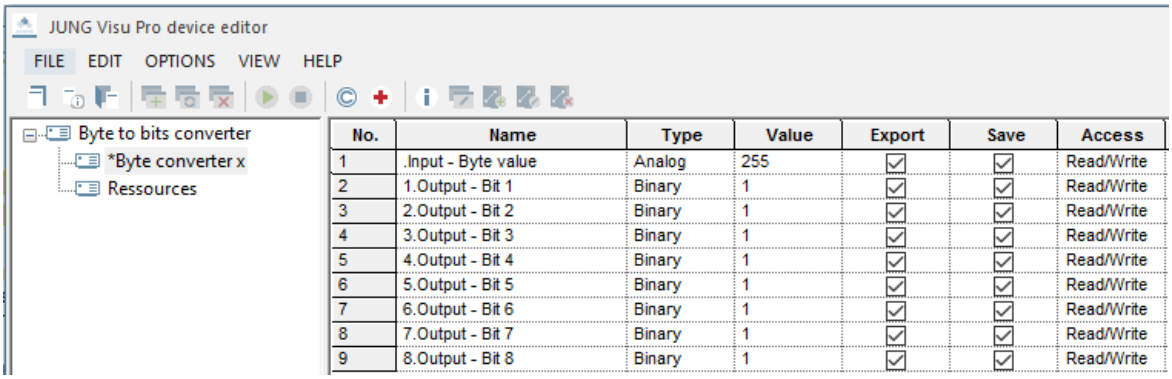
Several ***Byte to Bits Converter*** -***folders*** can be created. Rechtsklick auf den übergeordneten Knoten mit dem Namen des Prozessanschlusses öffnet ein Kontextmenü. Right-clicking on the parent node with the name of the process connection opens a context menu. Right-click to add further folders. The data point folder can be renamed, e.g. to designate the active site. The name of the folder has no effect on the processing of the script.

For each device folder further ***Byte to Bits Converter*** ***(*Byte converter x)*** may be created (max. 20 items). By right clicking further converters are added. The respective converter contains all data points required for conversion.

Data point folder of the Byte – Bits Conversion

To convert the value of the individual bits, the process interface requires a 1-byte variable as input. **The *Byte to Bits Converter* only processes unsigned data point types and a value range from 0...255.**

The significance (0 or 1) of the individual bits within the byte is returned to the output variables (output 1 to 8).



No.	Name	Type	Value	Export	Save	Access
1	.Input - Byte value	Analog	255	☑	☑	Read/Write
2	1.Output - Bit 1	Binary	1	☑	☑	Read/Write
3	2.Output - Bit 2	Binary	1	☑	☑	Read/Write
4	3.Output - Bit 3	Binary	1	☑	☑	Read/Write
5	4.Output - Bit 4	Binary	1	☑	☑	Read/Write
6	5.Output - Bit 5	Binary	1	☑	☑	Read/Write
7	6.Output - Bit 6	Binary	1	☑	☑	Read/Write
8	7.Output - Bit 7	Binary	1	☑	☑	Read/Write
9	8.Output - Bit 8	Binary	1	☑	☑	Read/Write

Figure 2

The input values of the Byte to Bits Conversion

Input: .Input – Byte value

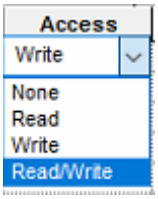
Input value (1 Byte; value range 0..255)

The output values of the Bye to Bits Conversion

Output: 1. Output – Bit 1 to 8. Output – Bit 8

Output value (1 Bit; value range 0 und 1)

Setting the access rights



The process interface script evaluates the access rights settings. Telegrams are only sent if the access rights are set to ***write*** or ***read/write***. In this script all access rights have been set optimally and should not be changed.

Data point folder resources

This folder contains no data.

3. Carrying out the process interface

Via the command ***Menu - Options - Start process interface*** or via the automatic start the process connection is started by the device editor.

The script is now ready for the conversion of a 1-byte variable into single bits. By sending a value to the input variable .***Input - Byte value*** the calculation operation in the script is started.Ausführung der Schreibbefehle

Execution of the write commands

Attention: As soon as the script was able to pass the telegram to the Device Editor, the command is considered executed and acknowledged to the process model. There is no check whether the data could actually be sent to the connected device and were received there!

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***write*** or ***read/write*** are set..

Execution of the read commands

If the ***read value*** command is executed by the process model or manually by the user, the current value of the data point is sent back to the process model as the result of the read operation

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***read*** or ***read/write*** are set.

System news

Via the tab ***Menu - View - Message Lists*** the telegram history or the input and output of the script is displayed. This allows the check during the runtime of the script.

Figure 3

JUNG Visu Pro
Device Editor

Process interface Min Max Value

Version 1.0

Inhalt:

Disclaimer 1
Contact person 2
Trademark 2
Functionality 3
1. Create the data point folder Input value x 3
2. Data point folder 3
2.1. Data point folder Input value x 3
2.2. Data point folder calculation 4
2.2.1. Setting the access rights 4
2.3. Data point folder resources 4
3.1 Execution ot the write commands 5
3.2 Execution ot the read commands 5

Disclaimer

No warranty is given for the correctness and functionality of the process interface. Liability for damages or consequential damages resulting from the use of the process interface is excluded.

This also applies to program components that do not originate from the author of the RS flip-flop itself.

By using this process interface the user accepts the disclaimer!

Contact person

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89
mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All rights reserved

Trademark

EIB®, KNX® and ETS® are registered trademarks of KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the USA and other countries.
OPC® is a registered trademark of the OPC Foundation.
Sax Basic Engine is a registered trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trade names, company names and product names are trademarks or registered trademarks of their respective owners.

Functionality

The process interface ***Min Max Value*** shows the minimum and maximum value of the values transferred to the process connection and calculates their average value.

Furthermore, the user is shown the time of the last update for each individual input value.

The process interface provides two types of folders for this purpose: A "Calculation" folder in which the results of the calculation are output and Data point folders that can be created by the user and act as input values for the calculation.

1. Create the data point folder Input value x

After creating the process interface, a default data point folder ***Input value x*** is displayed.

Several ***Input value x*** **- *folders*** can be created. A right click on the superordinate node with the name of the process connection opens a context menu. In the context menu, the creation of one or more further ***input value x*** data point folders can be selected.

The data point folder ***Input value x*** can be renamed, e.g. to indicate the origin of the value. The name of the folder has no effect on the processing of the script.

The ***Input value x*** folder contains all data points required for the calculation of minimum, maximum and average values and for controlling the script.

2. Data point folder

2.1. Data point folder Input value x

Input value

Current value of the input variable. The value is required for calculating the minimum, maximum and average value.

Timestamp

The time stamp records the time at which the value of the input variable received its last change.

Reset

The current value of the associated input variable is reset to "0".

2.2. Data point folder calculation

Minimum value

This field contains the minimum value determined during the calculation from all input values of this process connection.

Maximum value

This field contains the maximum value determined during the calculation from all input values of this process connection.

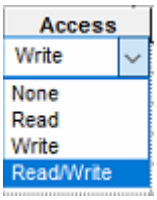
Average

This field contains the average value determined during the calculation from all input values of this process connection.

Reset

The current values of all input variables of this process connection are reset to "0“.

2.2.1. Setting the access rights



The process interface script evaluates the access rights settings. Telegrams are only sent if the access rights are set to ***write*** or ***read/write***. In this script all access rights have been set optimally and should not be changed.

2.3. Data point folder resources

This folder contains no data.

3. Carrying out the process interface

Via the command ***Menu - Options - Start process interface*** or via the automatic start the process connection is started by the device editor.

The script is now ready to calculate new minimum, maximum and average values. By sending an input value to the data point ***Input Value*** a new calculation process is triggered.

3.1 Execution ot the write commands

Attention: As soon as the script was able to pass the telegram to the Device Editor, the command is considered executed and acknowledged to the process model. There is no check whether the data could actually be sent to the connected device and were received there!

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***write*** or ***read/write*** are set.

3.2 Execution ot the read commands

If the ***read value*** command is executed by the process model or manually by the user, the current value of the data point is sent back to the process model as the result of the read operation

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***read*** or ***read/write*** are set.

JUNG Visu Pro
Device Editor

RS – Flipflop

Version 2.0

Content:

Disclaimer 1
Contact person 2
Trademark 2
1. Functionality 3
2. Create the data point folder *RS-Flipflop* 4
Data point folder RS-Flipflop 4
Setting the access rights 4
Data point folder Resources 4
3. Carrying out the process interface 5
Execution of the write commands 5
Execution of the read commands 5

Disclaimer

No warranty is given for the correctness and functionality of the process interface. Liability for damages or consequential damages resulting from the use of the process interface is excluded.

This also applies to program components that do not originate from the author of the RS flip-flop itself.

By using this process interface the user accepts the disclaimer!

Contact person

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1 D-58579 Schalksmühle
Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89
mail.info@jung.de

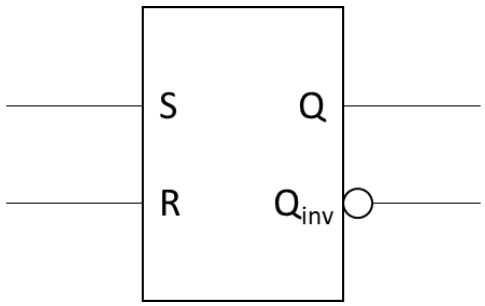
Copyright ©2019 ***Agentilo GmbH***
All rights reserved

Trademark

EIB®, KNX® and ETS® are registered trademarks of KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the USA and other countries.
OPC® is a registered trademark of the OPC Foundation.
Sax Basic Engine is a registered trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trade names, company names and product names are trademarks or registered trademarks of their respective owners.

1. Functionality

The ***RS-Flipflop*** process interface calculates the output variables [Q, Q_{inv}] depending on the two input variables [R,S]. The flip-flop is reset-dominated.

Input		Output		Circuit symbol
S	R	Q	Q_{inv}	
0	0	save		
0	1	1	0	
1	0	0	1	
1	1	1	0	

If there is a 1 signal at input [S] (Set) and a 0 signal at input [R] (Reset), the value 1 is output at output [Q]. This state continues until a 1 signal is present at the R input (Reset). Output [Q_{inv}] provides the inverted state.

If a 0 signal is present at input [S] and a 1 signal at input [R], the value 0 is output at output [Q]. Output [Q_{inv}] provides the inverted state.

If a 0 signal is present at input [S] and a 0 signal at input [R], the last value is maintained at output [Q]. Output [Q_{inv}] provides the inverted state.

If a 1 signal is present at input [S] and a 1 signal at input [R], output [Q] is set to 0 by the reset dominant flip-flop. Output [Q_{inv}] provides the inverted state.

2. Create the data point folder *RS-Flipflop*

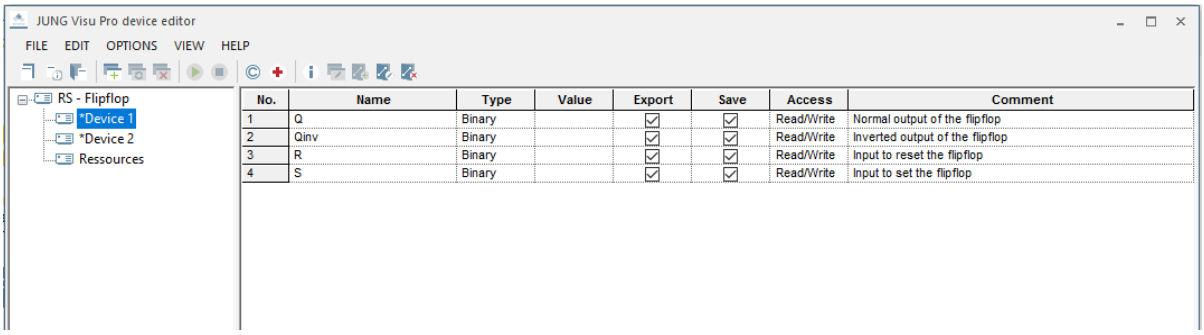
After creating the process interface a default data point folder RS-Flipflop is displayed in the directory structure.

Several ***RS-Flipflop*** - ***folders*** can be created. A right click on the superior node with the name of the process connection opens a context menu. A right click will add further folders. The datapoint folder can be renamed, e.g. to designate the place of action. The name of the folder has no effect on the processing of the script.

Further ***RS-Flipflop*** - ***limbs*** can be created per device folder. Right click on the device folder opens a context menu. A right click adds further flip-flops. The flip-flop contains all data points required for control.

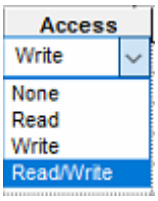
Data point folder RS-Flipflop

To calculate the flip-flop the input [R] is required. Afterwards the two outputs Q and Q_{inv} are output.



No.	Name	Type	Value	Export	Save	Access	Comment
1	Q	Binary		☑	☑	Read/Write	Normal output of the flipflop
2	Qinv	Binary		☑	☑	Read/Write	Inverted output of the flipflop
3	R	Binary		☑	☑	Read/Write	Input to reset the flipflop
4	S	Binary		☑	☑	Read/Write	Input to set the flipflop

Setting the access rights



The process interface script evaluates the access rights settings. Telegrams are only sent if the access rights are set to ***write*** or ***read/write***. In this script all access rights have been set optimally and should not be changed.

Data point folder Resources

This folder contains no data.

3. Carrying out the process interface

Via the command ***Menu - Options - Start process interface*** or via the automatic start the process connection is started by the device editor.

Execution of the write commands

Attention: As soon as the script was able to pass the telegram to the Device Editor, the command is considered executed and acknowledged to the process model. There is no check whether the data could actually be sent to the connected device and were received there!

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***write*** or ***read/write*** are set.

Execution of the read commands

If the ***read value*** command is executed by the process model or manually by the user, the current value of the data point is sent back to the process model as the result of the read operation.

The access rights setting is evaluated by the script. The current value is only returned if the access rights ***read*** or ***read/write*** are set.

JUNG Visu Pro
Device Editor

Process Interface “Bidirectional UDP Communication”

Version 1

Content:

DISCLAIMER....2

FUNCTIONALITY....4

1. Specification of the Local Communication Port....4
2. Datapoints of an IP-Capable Device....5
   - 2.1. Device Folder....5
   - 2.2. Creation of Datapoint Folders....6
     - 2.2.1. Datapoint Folder *Switching Command*....6
     - 2.2.2. Datapoint Folder *Analog Command*....8
     - 2.2.3. Datapoint Folder *Date/Time*....9
     - 2.2.4. Datapoint Folder *Text*....10
     - 2.2.5. Using Numerical Values in Command Formats....11
     - 2.2.6. Access Control....11
3. Running the "Bidirectional UDP Communication" Process Interface....12
   - 3.1 Execution of Write Commands....12
   - 3.2 Execution of Read Commands....12

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "Bidirectional UDP communication" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "Bidirectional UDP communication" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "Bidirectional UDP communication" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

Functionality

The process interface "Bidirectional UDP communication" allows to send and receive UDP data packets over the Internet to IP-capable devices, with each an own IP address. UDP ("User Datagram Protocol") is fundamental to the IP network and does not presume a certain structure of the UDP data packets.

The contents and meaning of the UDP data packets depend on the specific device, but the user can specify the contents of the data packets in order to implement binary switch, analog set, date/time set and text commands and corresponding data points. These datapoints reside in the datapoint folders.

Moreover, sending UDP data packets requires a local communication port. This port and similar information resides in the resources folder.

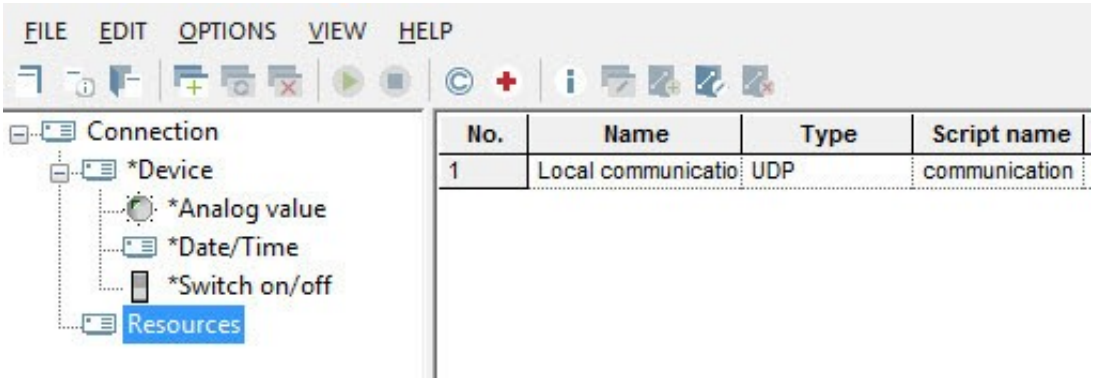
When using the "Bidirectional UDP communication" software, it may be necessary to configure firewalls, router and other devices to enable the sending / receiving of UDP telegrams over the network by the software. These measures are the user's responsibility and are not covered here.

1. Specification of the Local Communication Port

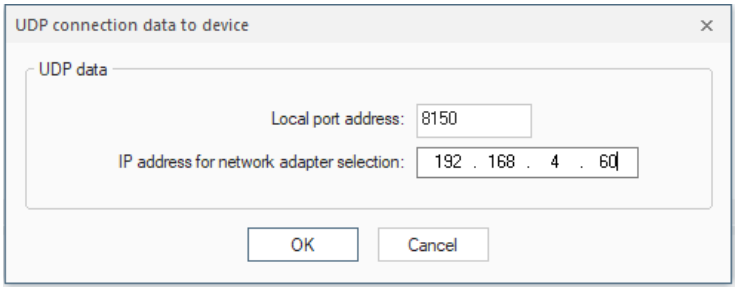
As part of the Internet data protocol, most notably the Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP), a number is used to distinguish concurrent communications between the endpoints of an IP connection. The endpoint is called a port, the number is called the port number. With different ports, a computer can virtually at the same time connect to a web server, receive e-mails, and use or provide other services.

Some port numbers are reserved for special purposes, for example the port number 80 is used for the HTTP protocol.

For the "Bidirectional UDP communication" software, a free port number must be found and assigned. It can be specified in the resources folder.
Click on the resource tree item to find the properties of the UDP communication port in the resources data table.



Click the button in the row ***communication port***, column ***Edit*** to see or change the port number used.



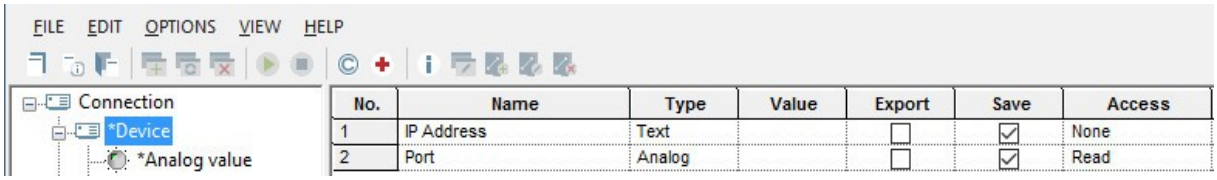
The port number should not be used by another application and its status should switch from 'closed' to 'open' when the process interface starts. It will remain open until the process interface is stopped. The status is visible in the column ***value*** in the row of the communication port.

The name of the resource folder cannot be changed.

2. Datapoints of an IP-Capable Device

After a process interface of the "Bidirectional UDP communication" type has been created, a new tree node with the name ***device*** is added. The node represents an IP-capable device. From hereafter we will call this node the "device node", though in the actual project it should be renamed to something more descriptive, for example "beamer", "remote control" or "sound system" or "sound system volume" depending on the device and application.
Note that different command/datapoint sets of one device or different devices may be described each in different datapoint folders.

The asterisk in front of the node name signals that more than one node of this kind may be created by right-clicking its tree parent node and opening the context menu.



2.1. Device Folder

Basic information regarding the device includes its IP address in text form, for example 192.168.0.110. If an IP name has been assigned, which resolves to this address, the name can be used instead of the address.

In addition to the address, a port number must be specified to properly define the recipient of the UDP data packet.

With the port specified in the resources folder (and implicitly the computer IP address) for the sender and both, IP address and port of the IP-capable device, as the receiver, both end points of the IP communication are defined.

The device folder is created including already the subfolders ***Analog value***, ***Date/Time***, ***Switch on/off, Text*** datapoint folders for possible commands and corresponding datapoints of the respective type.

2.2. Creation of Datapoint Folders

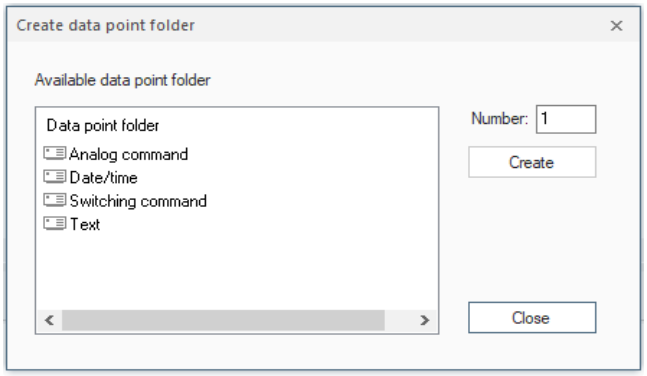
Each datapoint of a device is represented with a datapoint folder inside of the device folder. When the data point folder is selected with a left click, a table of the datapoint properties is displayed.

In order to define more data points and in addition to the default datapoint folders, more datapoint folders can be created as subitems to the device folder.

To create a new datapoint folder, select the device folder in the structure tree, then click the right mouse button to open its context menu. In the context menu select the command ***Create datapoint folder***. Alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create data point folder*** opens. Select the desired datapoint type and click the button ***Create***.

The datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.



2.2.1. Datapoint Folder *Switching Command*

This folder contains five data points:

- **Binary value**

  This data point is of the type 'binary' and is exported per default to the process model. Exporting means that it will be visible in the process model as a regular binary datapoint. It will be visible in the visualization editor if the process model exports the data point as well.

  When the datapoint is switched off, the script will send the text for entry 'Command value OFF'. Switching on is similar: the text for entry 'Command value ON' will be sent to the device.

- **Command value OFF**

  The value of this variable is the text which will be sent for the logical state 'OFF'.

- **Command value ON**

  The value of this variable is the text which will be sent for the logical state 'ON'.

- **Feedback value OFF**

  The value of this variable is the text which has to be received to set the ***Binary value*** 'OFF'.

- **Feedback value ON**

  The value of this variable is the text which has to be received to set the ***Binary value*** 'ON'.

Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember how to turn on the device.

If the export flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
Per default the export flag is not set.

2.2.2. Datapoint Folder *Analog Command*

This folder contains three data points:

- **Analogue Value**

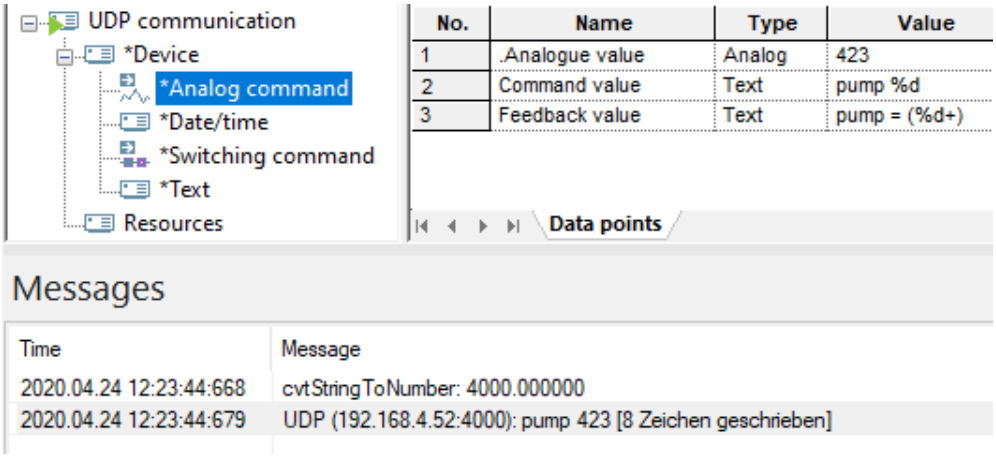
  This datapoint is representing an analog value. It is exported per default to the process model where it is a regular analog data point.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the 'Command value' entry.

- **Command value**

  The 'Command value' entry defines the command string to be sent to the device when the value of the analog datapoint is changed in the visualization.

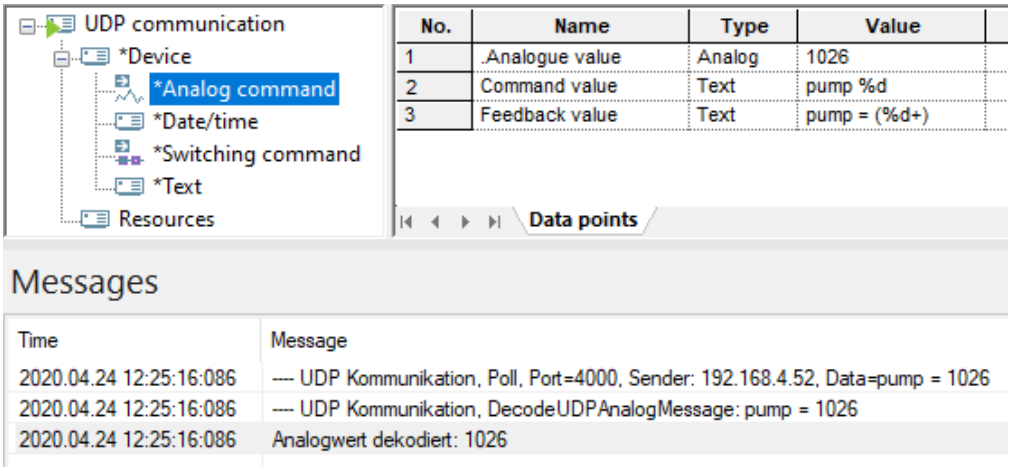
  In the command string the actual value to be sent is represented by the %d placeholder. Before sending, the script will replace %d with the actual value.



- **Feedback value**

  The 'Feedback value' entry defines the command string which has to be received to set the ***Analogue value***.

  In the command string of the UPD data packet the actual value to be sent to the ***Analogue value*** is represented by the (%d+) placeholder.



Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

If the export flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
Per default, the export flag is not set.

2.2.3. Datapoint Folder *Date/Time*

This folder contains two data points:

- **Date/Time**

  The type of this data point is 'Date/Time'. Per default, it will be exported to the process model.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the 'Command Text' entry.

- **Command value**

  The 'Command value' entry defines the command string to be sent to the device when the value of date/time datapoint is changed in the visualization.

  In the command string the actual value to be sent is represented by the %s placeholder. Before sending, the script will replace %s with the actual value.

  Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

  If the export flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
  Per default, the export flag is not set.

2.2.4. Datapoint Folder *Text*

This folder contains three data points:

- **Text Value**

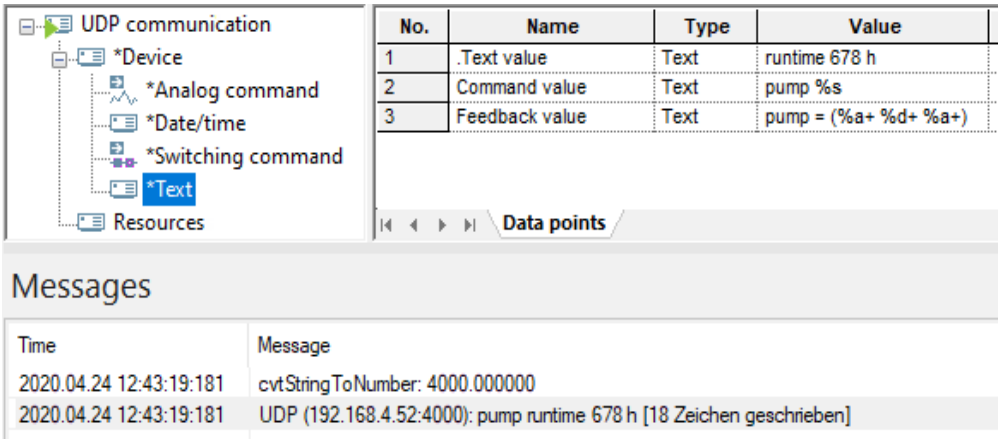
  This datapoint is representing a text value. It is exported per default to the process model where it is a regular string data point.

  If the value of this datapoint is changed in the visualization, in consequence the script sends a text formatted according to the 'Command value' entry.

- **Command value**

  The 'Command value' entry defines the command string to be sent to the device when the value of the text datapoint is changed in the visualization.

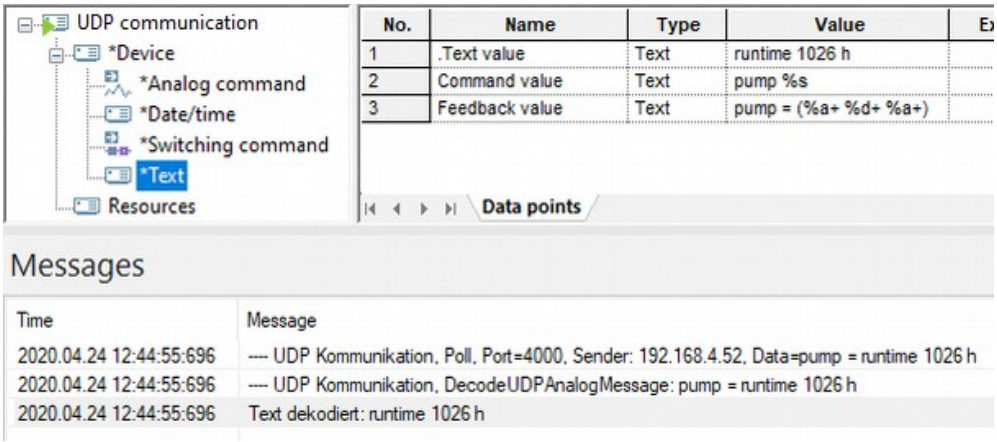
  In the command string the actual value to be sent is represented by the %s placeholder. Before sending, the script will replace %s with the actual value.



- **Feedback value**

  The 'Feedback value' entry defines the command string which has to be received to set the ***Text value***.

  In the command string of the UPD data packet the actual value to be sent to the ***Text value*** is represented by the (%a+) placeholder for alphabetic characters and the (%d+) placeholder for numbers.



Important: The *save* flag must be set to remember this command text in the devices project! Otherwise, the editor will not remember the command format and will not be able to build the command string to be sent.

If the export flag has been set, the variable will be exported. As a result, it is possible to edit the command text in the visualization editor!
Per default, the export flag is not set.

2.2.5. Using Numerical Values in Command Formats

In command formats it is possible to enter characters as numerical values. A substring (called "escape sequence") in format \ddd, where ddd must be a 1 – 3 digit value between 1 and 255, will be transmitted as a byte with this value.

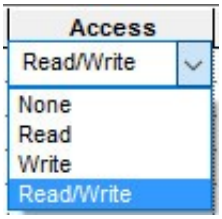
Examples:

\10 encodes a linefeed
\64 encodes the at sign '@'

Note that a byte with the value 0 cannot be used.
If the char '\' is to be inserted literally, it must be duplicated.

Format "\\001" will send "\001" whereas "\001" will send the byte with value 1.

2.2.6. Access Control



The script evaluates access rights. UDP datagrams will only be sent when access rights ***Write*** or ***Read/Write*** are specified for the datapoint.

3. Running the "Bidirectional UDP Communication" Process Interface

To start the "Bidirectional UDP communication" process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started. In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device succeeds, the script will recognize changes carried out on the datapoints in the process model respectively the visualization. Then the script checks access rights and builds the strings to be sent. Finally, the script transfers these telegrams to the devices editor, which actually performs the sending.

3.1 Execution of Write Commands

Important: Transferring the UDP datagrams to the devices editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device!

The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

3.2 Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no UDP telegram will be created. Instead, the current value of the data point which is the value last sent, will be reported back.

The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
Device-Editor

Prozessinterface
Simple TCP Client Communication

Version 1

Content:

Disclaimer **2**

Functionality **4**

1. Specification of the Server Communication **5**

2. Data point folder **6**

2.1. Creation of Datapoint Folders 6
2.1.1. Folder Switch Command 7
2.1.2. Folder Analog Command 8
2.1.3. Folder Date/ Time 10
2.1.4. Folder Text 11
2.1.5. Using Numerical Values in Command Formats 12
2.2. Access Control 12

3. Running the Process Interface **13**

3.1. Execution of Write Commands 13
3.2. Execution of Read Commands 13

Disclaimer

Reasonable effort is made to maintain the currency and accuracy of this document and the correct implementation of the "Simple TCP Client Communication" software described here, but the use of the information and the software is the sole responsibility of the user. We do not accept any liability for the up-to-dateness, correctness, completeness, suitability or quality of both document and software.

The software is developed with great care, but in no event we warrant that the software is error free or that the customer will be able to operate the software without problems or interruptions. We do not warrant for problems due to improper use of the software or other causes, which are out of the sphere of our influence.

The user must assume the entire risk of using the software. In no event we will be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the software, even if we have been advised of the possibility of such damages.

This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall we or our suppliers' liability to the customer, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the customer.

The "Simple TCP Client Communication" software is a sample provided with source code, that you can modify according your needs and in your responsibility.

If you do not agree to these terms, you are not allowed to use the "UDP commands" software.

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

Functionality

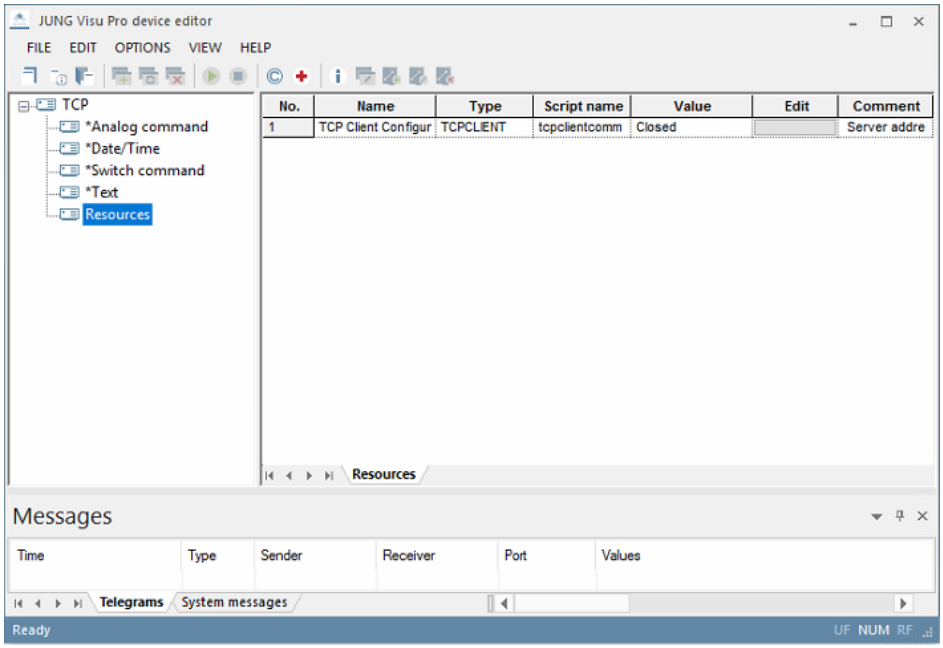
The process interface **Simple TCP client communication** enables Data exchange of TCP data packets ('Transmission Control Protocol') with a TCP Server that can be accessed via an IP address.
The user can use the process interface **Simple TCP client communication** for switching commands, analog values, time information or text telegrams, which are then sent by the device editor via TCP protocol over a network or are received as value messages.

The process interface provides two types of folders for this purpose: An Resource folders for entering the server data and data point folders that are created by the users can be created.

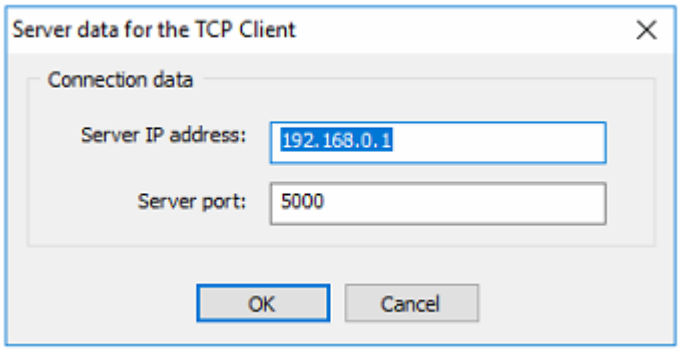
The datapoint folders allow you to create, edit and delete a set of Data points on which communication with the server takes place.

1. Specification of the Server Communication

For communication with a TCP server, the process interface requires the IP address and the port of the server.
The resource folder can be selected by clicking on it in the directory tree. In the corresponding display window on the right side, a table appears in which the corresponding settings must be made.



Click the button in the ***Edit*** column to open a dialog for entering the IP address and port.



After the process interface is started, the port for sending the TCP telegrams is opened. The port remains open until the process interface is stopped.
At running process interface, the current state of the port (open/closed) is displayed in of the ***Value*** column is displayed.
The name of the resource cannot be changed.

2. Data point folder

2.1. Creation of Datapoint Folders

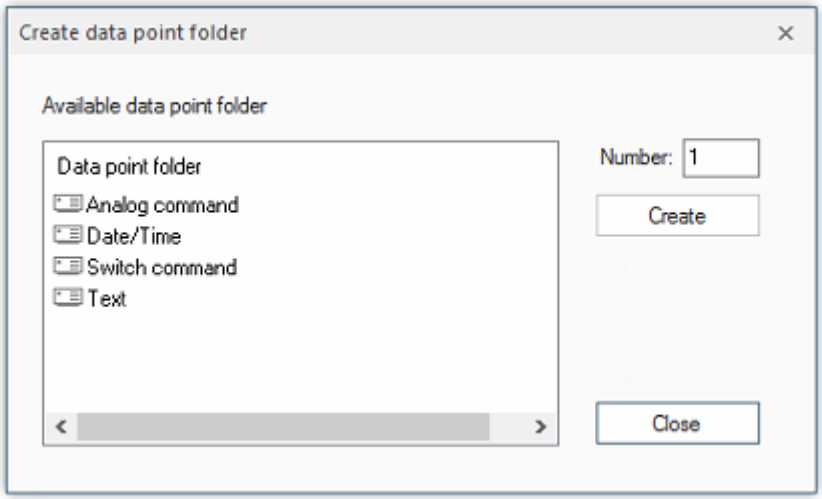
Each datapoint of a device is represented with a datapoint folder inside of the device folder. When the data point folder is selected with a left click, a table of the datapointproperties is displayed.

Analogue command, ***date/time***, ***switching command*** and ***text*** are available as data point or folder types.

In order to define more data points and in addition to the default datapoint folders, more datapoint folders can be created as subitems to the device folder.

To create a new datapoint folder, select the device folder in the structure tree, then click the right mouse button to open its context menu. In the context menu select the command ***Create datapoint folder***. Alternatively, choose the menu item ***Edit*** and click ***Create datapoint folder***.

The dialog ***Create data point folder*** opens. Select the desired datapoint type and click the button ***Create***.



The datapoint folders may be renamed at any time. This has no impact on the working of the script, because the script uses internal names which are fixed.

Note: When the process interface is started, no new data points can be can be created.

2.1.1. Folder Switch Command

The ***switch command*** data point folder contains 5 data points:

- **Switch command**

  This data point is of the binary type and is sent to the Process model of the visualization exported. It is exported there as a normal binary process variable is used.

  If this data point is **switched off** by the visualization, then the script of the process interface uses the text that is stored as value under ***Switching command OFF*** is entered as TCP telegram to the server.

  If this data point is **switched on** by the visualisation, then the script of the process interface uses the text that is stored as value under ***Switch command ON*** is entered as TCP telegram to the server.

  If the process connection sends a TCP telegram with the text entered as value is entered under ***value message OFF,*** the script of the process interface the value **switched off** at this data point and thus for visualization.

  If the process connection sends a TCP telegram with the text that is stored as value is entered under ***value message ON***, the system sends the script of the process interface the value **switched on** at this data point and thus for visualization.

  ➢ **Important**: The ***save*** flag should always be set, otherwise this text is lost after closing the Device Editor!

  If the ***export*** flag is set, this data point is also transferred to the process model and exported there as a normal data point treated.

  This makes it possible, for example, to edit this text even in the visualization server. The ***export*** flag is not set by default.

2.1.2. Folder Analog Command

The ***analog command*** data point folder contains 3 data points:

- **Analog command**

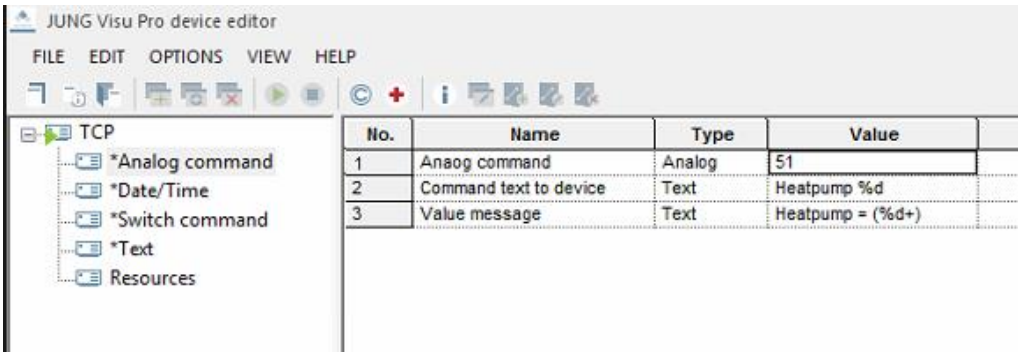
  This data point is of type Analog and is connected to the process model of the visualization exported. It is exported there as a normal analog process variable is used.
  If the value of this data point is changed by the visualization the script of the process interface sends the text, which is stored as value under command text is entered on the device, as a TCP telegram to the server.

- **Command text to device**
-

  The value of this variable is the text that is sent to the server when the value of the variable ***Analog command*** is changed by the visualization.

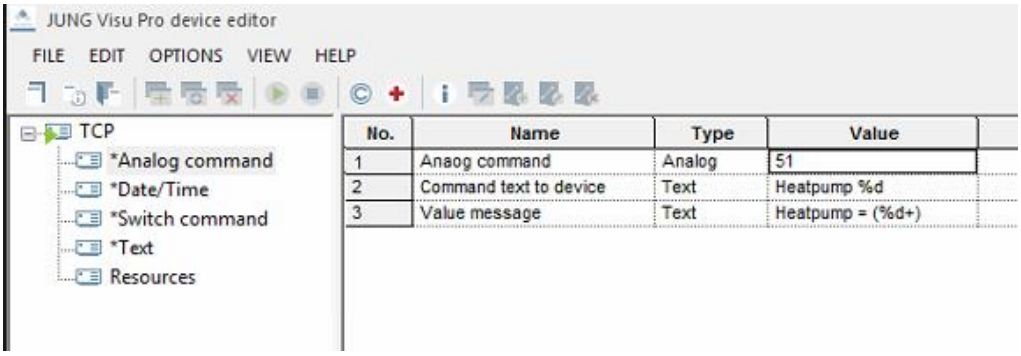
  The current value can be accessed in the text with ***%d.***

  The script replaces ***%d*** in the text with the current value of the ***Analog command*** variable.



- **Value report**
  The value of this variable is the text that must be sent by the TCP server to change the value of the ***Analog command*** variable by the TCP server in the visualization.

  The current value in the TCP telegram can be accessed with ***(%d+).*** The script passes the current message of the TCP telegram to the ***Analog command*** variable.



Important: The save ***flag*** should always be set, otherwise this text will be lost after closing the Device Editor!
If the ***export*** flag is set, this data point is also exported to the process model and treated as a normal data point. This makes it possible, for example, to edit this text even in the visualization server. The ***export*** flag is not set by default.

2.1.3. Folder Date/ Time

The ***Date / Time*** data point folder contains 2 data points:

- **Date / Time**

  This data point is of the type ***date/time*** and is sent by default to the process model of the visualization is exported. It is exported there as a normal time process variable is used.
  If the value of this data point is changed by the visualization the script of the process interface sends the text, which is stored as value under ***command text*** is entered on the device, as a TCP telegram to the server.

- **Command text to device**
-

  The value of this variable is the text that is sent to the server, if the value of the variable Date/Time has been changed by the visualization is changed.

  You can access the current value of the time variable in the text with ***%s*** can be accessed. The script replaces ***%s*** in the text with the current value of the variable ***date/time***.

  Important: The save ***flag*** should always be set, otherwise this text will be lost after closing the Device Editor!
  If the ***export*** flag is set, this data point is also exported to the process model and treated as a normal data point. This makes it possible, for example, to edit this text even in the visualization server. The ***export*** flag is not set by default.

2.1.4. Folder Text

The ***Text*** data point folder contains 3 data points:

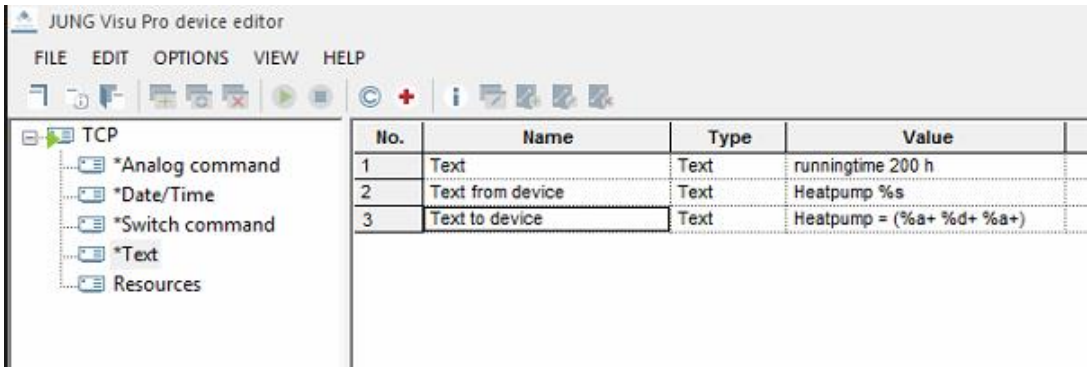
- **Text**

  This data point is of type text and is passed by default to the Process model of the visualization exported. It is exported there as a normal string process variable is used.
  If the value of this data point is changed by the visualization the script of the process interface sends the text, which is stored as value under ***command text*** is entered on the device, as a TCP telegram to the server.

- **Command text to device**
-

  The value of this variable is the text that is sent to the server, if the value of the variable text has been changed by the visualization is changed.

  You can access the current value of the text variable in the text with ***%s*** can be accessed. The script replaces ***%s*** in the text with the current value of the variable ***Text***.



- **Command text from device**
-

  The value of this variable is the text sent by the server to change the value of the Text variable in the visualization. You can access the current value in the TCP telegram with ***(%a+),*** for letters, and ***(%d+),*** for numbers. The script passes the current value of the TCP telegram to the Variable ***text***.

  Important: The save ***flag*** should always be set, otherwise this text will be lost after closing the Device Editor!
  If the ***export*** flag is set, this data point is also exported to the process model and treated as a normal data point. This makes it possible, for example, to edit this text even in the visualization server. The ***export*** flag is not set by default.

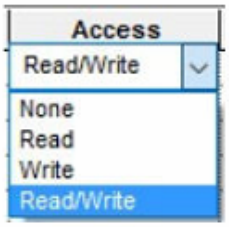
2.1.5. Using Numerical Values in Command Formats

In command formats it is possible to enter characters as numerical values. A substring (called "escape sequence") in format \ddd, where ddd must be a 1 – 3 digit value between 1 and 255, will be transmitted as a byte with this value.

Examples:
\10 encodes a linefeed
\64 encodes the at sign '@'

Note that a byte with the value 0 cannot be used.If the char '\' is to be inserted literally, it must be duplicated. Format "\\001" will send "\001" whereas "\001" will send the byte with value 1.

2.2. Access Control



The script evaluates access rights. HTTP GET requests will only be sent when access rights Write or Read/Write are specified for the datapoint.

3. Running the Process Interface

To start the " Simple TCP Client Communication " process interface, choose the menu item ***Options*** and click ***Start process interface*** or it will be started automatically when configured and the devices project is started.
In consequence, it may be started automatically from the process model.

After the initialization and if the connection to the device succeeds, the script will recognize changes carried out on the datapoints in the process model respectively the visualization. Then the script checks access rights and builds the HTTP GET requests to be sent. Finally, the script transfers these requests to the device editor, which actually performs the sending.

3.1. Execution of Write Commands

Important: Transferring the HTTP GET requests to the device editor is the last operation of the script regarding sending. Afterward, execution of the command is reported as successful to the process model. There will be no check whether the transfer has been successful and the data have been received by the IP-capable device! The new value sent will be reported back to the process model if the access rights ***Write*** or ***Read/Write*** have been set.

3.2. Execution of Read Commands

When a "Read command" is invoked by the process model or the user, no HTTP GET requests will be created. Instead, the current value of the datapoint which is the value last sent, will be reported back. The current value will be reported back only if the access rights ***Read*** or ***Read/Write*** have been set.

JUNG Visu Pro
vitaLED Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....8

4 FUNCTIONS....8

4.1 Project Organizer....8
4.1.1 Create vitaLED Project....9
4.1.2 Delete vitaLED Project....10
4.1.3 Rename vitaLED Project....10
4.1.4 Open vitaLED Project....11
4.2 Edit Project Properties....11
4.3 vitaLED Settings....12
4.4 Installation Settings....12
4.5 Connect to vitaLED Servers....13
4.6 Send Telegrams Manually....13
4.7 Sequences....15
4.8 Message List....16
4.9 Logging vitaLED Messages into Archives....17

5 REPORTS....18

5.1 Create Report....18
5.2 Print Preview....20

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® und ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

The RGBW ***vitaLED*** lighting solution of BRUMBERG is a lighting system for intelligent lighting to regulate steplessly the whole spectrum of white light range of 2.000 to 10.000 Kelvin in addition to the RGB color mode.

That offers not only the possiblity to immerse rooms in different ambiences by light colors and color gradients. The advantage of ***vitaLED*** is the support of the natural power curve consistent with the human circadian rhythm. The promotion of performance and wellness has been proven in several studies.

After the installation of the visualization system a program group is created in the Start Menu. The default settings can be adopted or changed manually during installation.

Click the program symbol to start the control panel of the visualization system. In order to clearly distinguish between the different applications, different colors were used. The icon of the **vitaLED editor** contains a blue colored bar .

To start the individual function modules, double-click the symbol of the respective modules.

The **vitaLED editor** is used to create vitaLED projects which serve as a basis for the communication with the vitaLED servers of lighting control.

First steps:

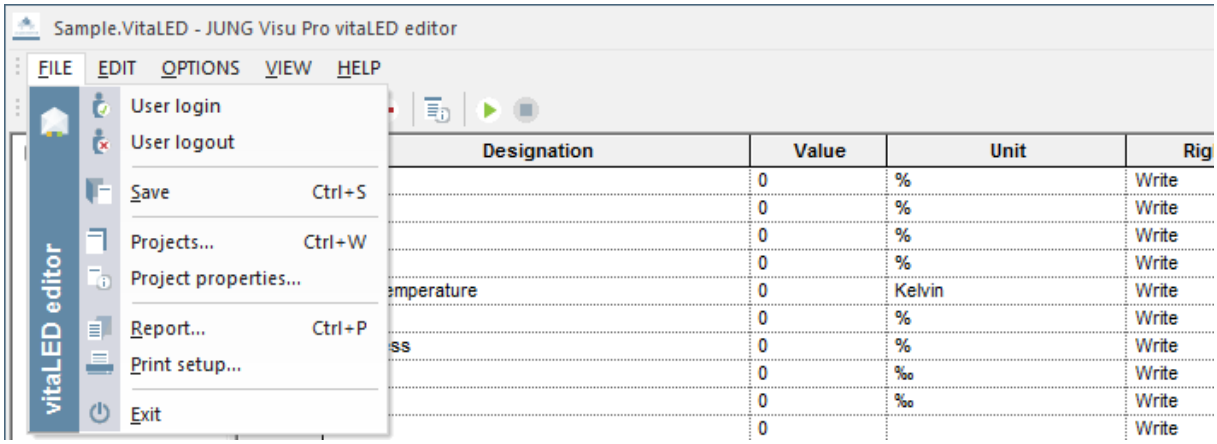
1. Create a vitaLED project.
2. Create an installation.
3. Make the desired entries, for example name of the installation, IP address, user name and password.
4. Save the settings.
5. Connect the project to the vitaLED bus.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

3 User Interface

3.1 Menu

Menu item File



User login: Opens a dialog to log in a user to the system.

User logout: Logs current user out of the system.

Save: Saves the project.

Projects: Opens a dialog to create and maintain vitaLED projects.

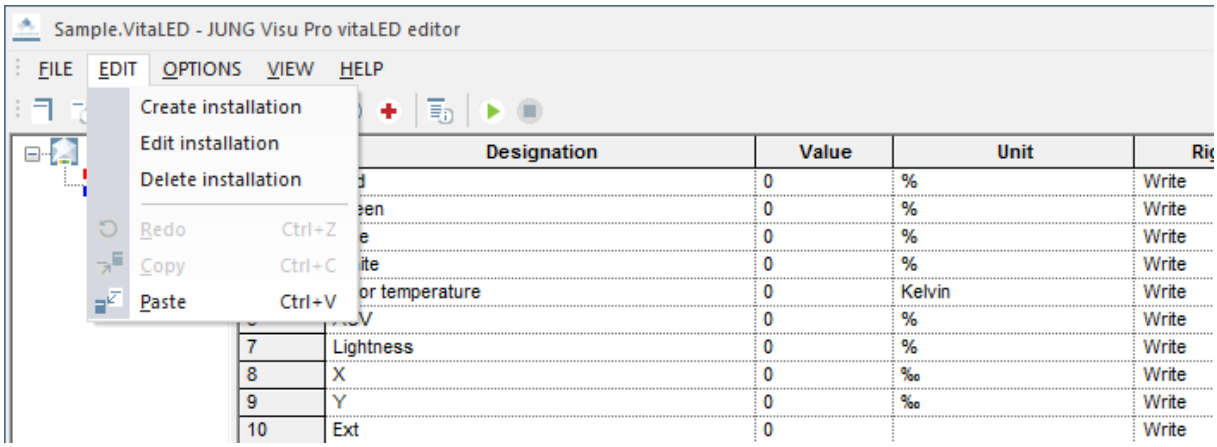
Project properties: Opens a dialog where you can edit general project properties.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Quits the vitaLED editor. Shortcut ALT+F4.

Menu item Edit



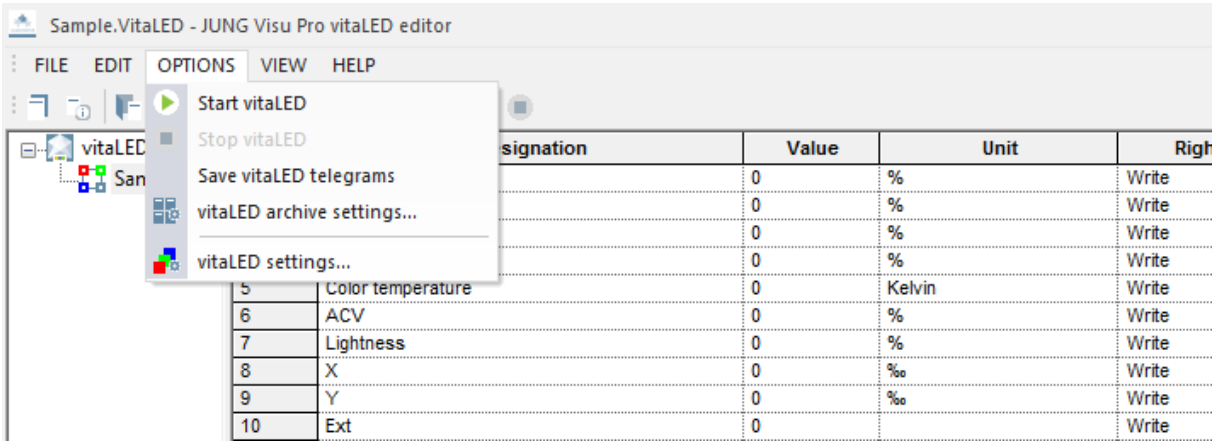
Create installation: Opens the dialog ***vitaLED installation properties***.

Edit installation: Opens the dialog for editing the properties of the selected vitaLED installation.

Delete installation: Deletes the selected vitaLED installation.

Edit: Opens the dialog for editing an existing project by standard options such as Undo, Copy, Paste or Delete.

Menu item Options



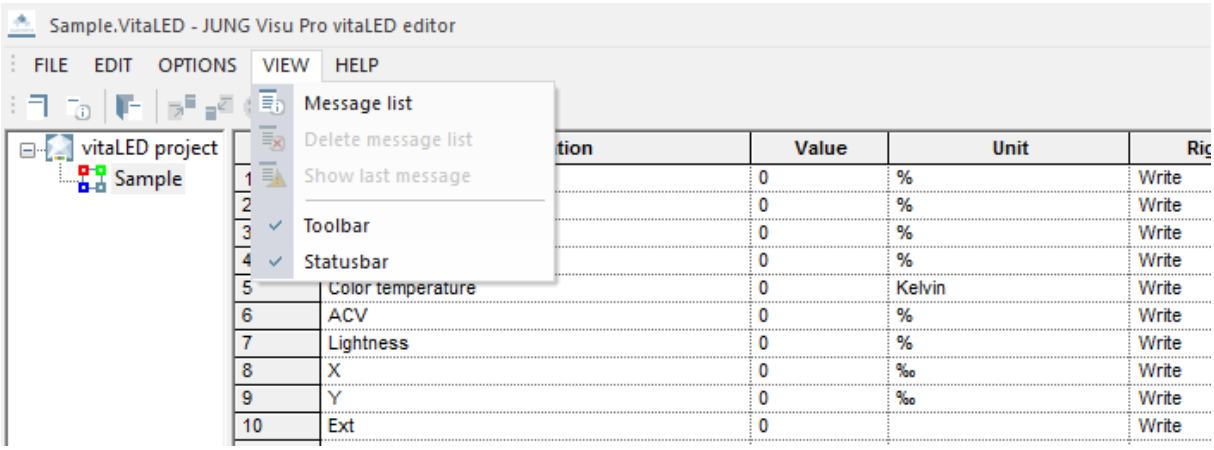
Stop / Start vitaLED: Connects to or disconnects from the servers of vitaLED.

Save vitaLED telegrams: Saves the vitaLED telegrams.

VitaLED archive settings: Opens the dialog ***Message archive settings***.

VitaLED settings: Opens the dialog ***vitaLED settings***.

Menu item View



Message list: Toggles the visibility of the messages list at the bottom of the editor.

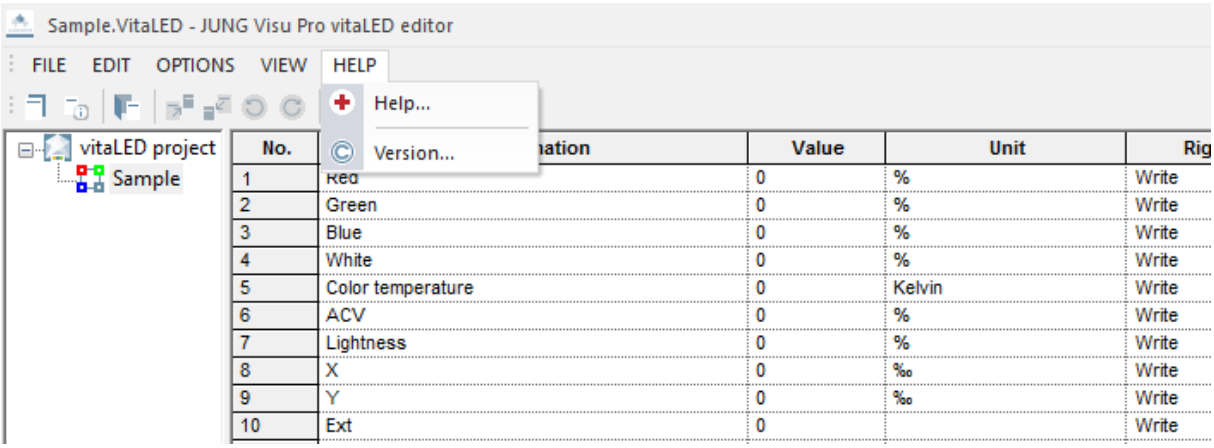
Delete Message list: Deletes the messages from the message window. This operation does not affect the recording of telegrams to a log file.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

Toolbar: Hides or shows the toolbar.

Statusbar: Hides or shows the status bar.

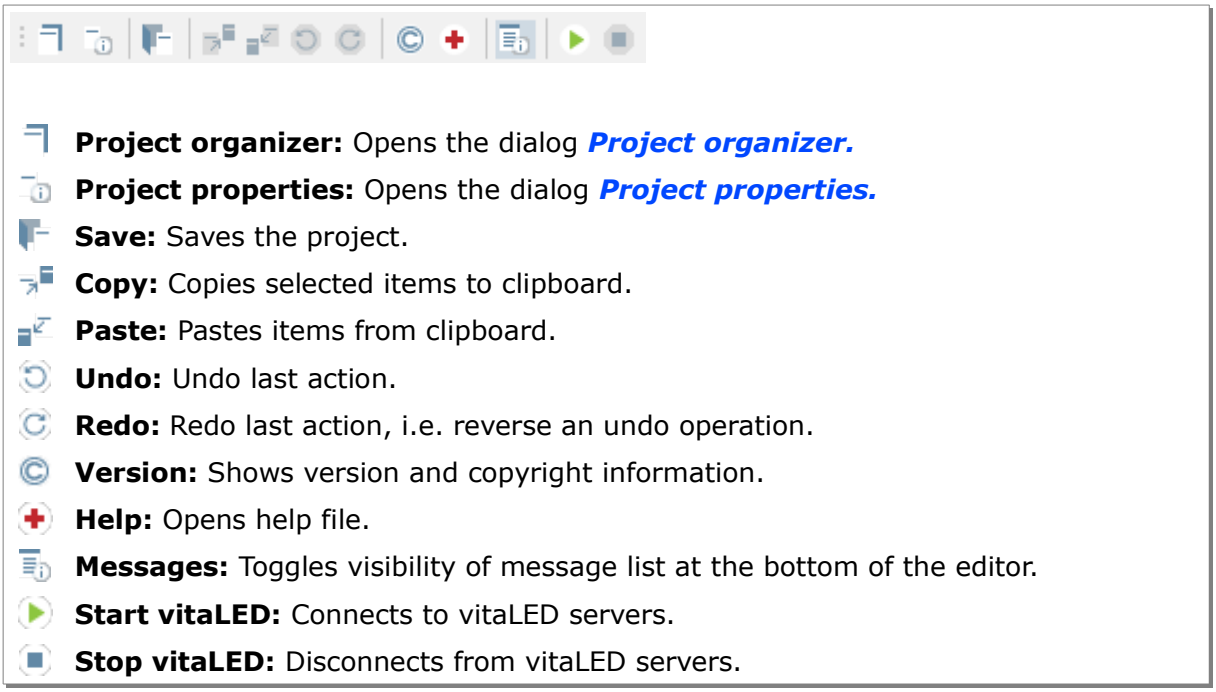
Menu item Help



Help: Opens help file.

Version: Shows version and copyright information.

3.2 Toolbar



Project organizer: Opens the dialog ***Project organizer.***
Project properties: Opens the dialog ***Project properties.***
Save: Saves the project.
Copy: Copies selected items to clipboard.
Paste: Pastes items from clipboard.
Undo: Undo last action.
Redo: Redo last action, i.e. reverse an undo operation.
Version: Shows version and copyright information.
Help: Opens help file.
Messages: Toggles visibility of message list at the bottom of the editor.
Start vitaLED: Connects to vitaLED servers.
Stop vitaLED: Disconnects from vitaLED servers.

4 Functions

4.1 Project Organizer

In the dialog ***vitaLED project organizer*** you can create new projects and open, close, delete or rename existing projects.

In the project organizer you can use the buttons on the sidebar or the commands under the menu item ***Project***.

The project organizer provides the following options:

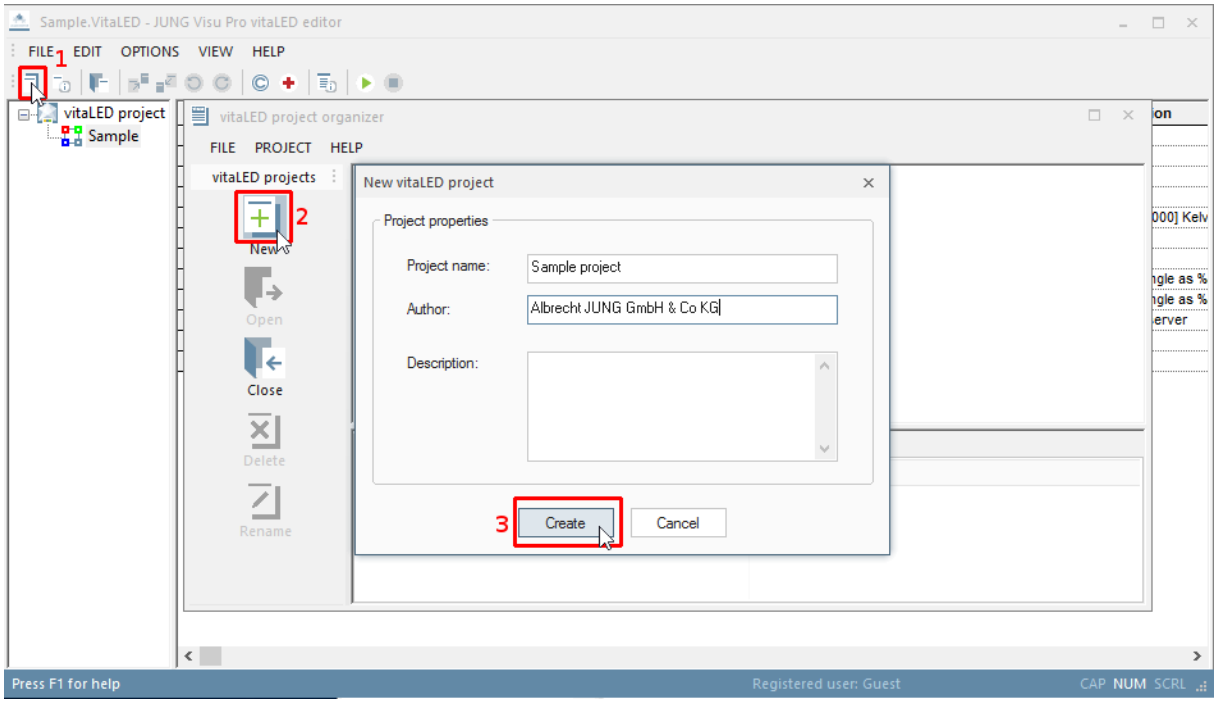
- **New**: Create a new vitaLED project.
- **Open**: Open an existing project in the vitaLED editor.
- **Close**: Close the currently opened vitaLED project.
- **Delete**: Delete the selected project.
- **Rename**: Rename a project.

4.1.1 Create vitaLED Project

The dialog ***vitaLED project organizer*** opens automatically when the vitaLED editor is started.

To start it manually in the vitaLED editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

To create a new vitaLED project, click the button ***New*** on the sidebar or choose the menu item ***Project*** and click ***New***. The dialog ***New vitaLED project*** opens.



Enter the data of the new project.

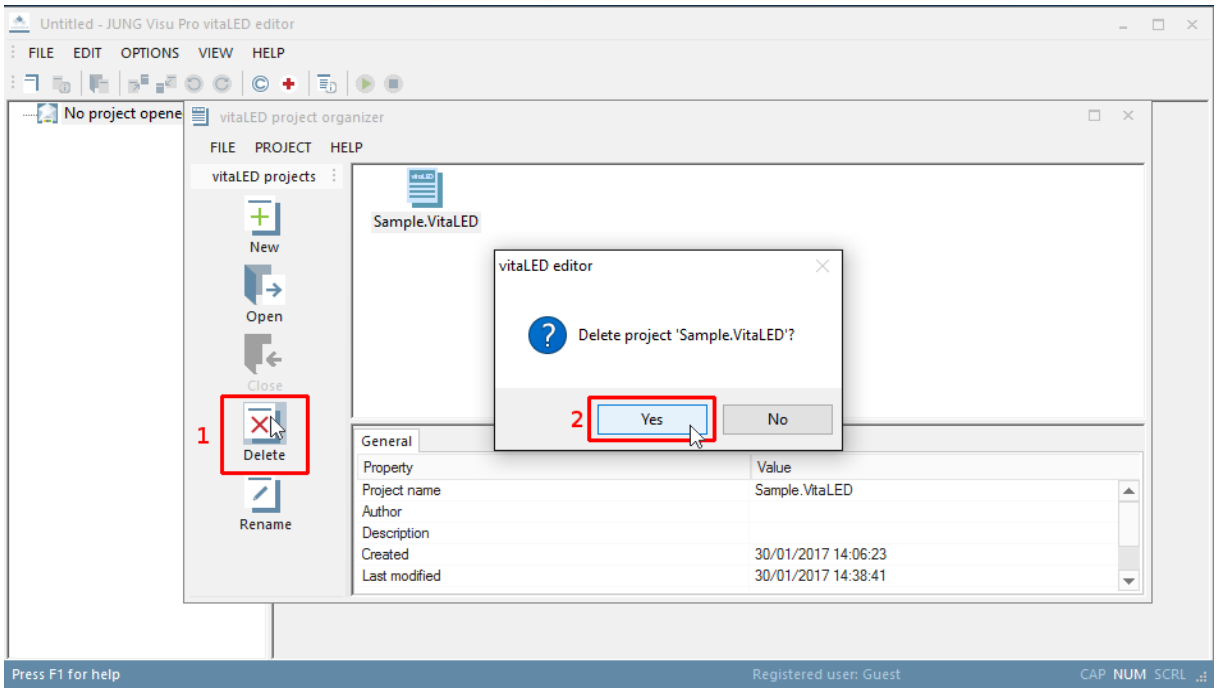
Project name: Enter a unique name for the vitaLED project.

Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or edited later.

Create: Press the button ***Create*** to commit the entries and to finish the creation of the project. The newly created project is now available in the project organizer.

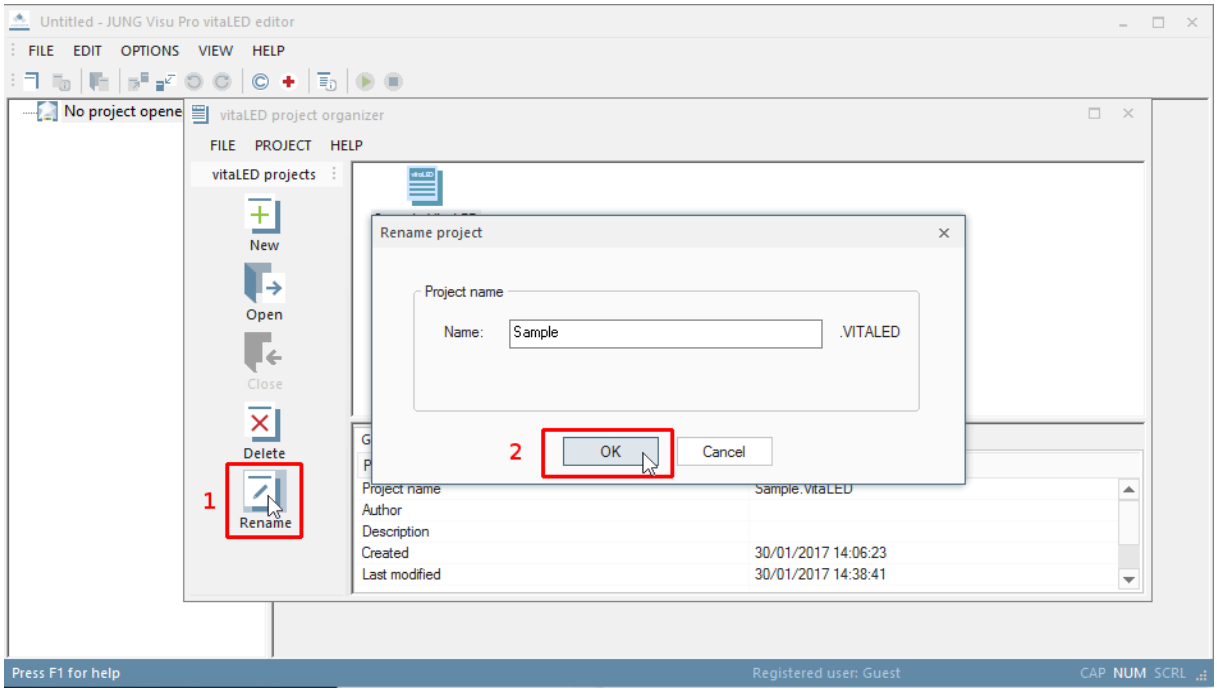
4.1.2 Delete vitaLED Project

To delete a project, select the project and click the button ***Delete*** in the sidebar.



4.1.3 Rename vitaLED Project

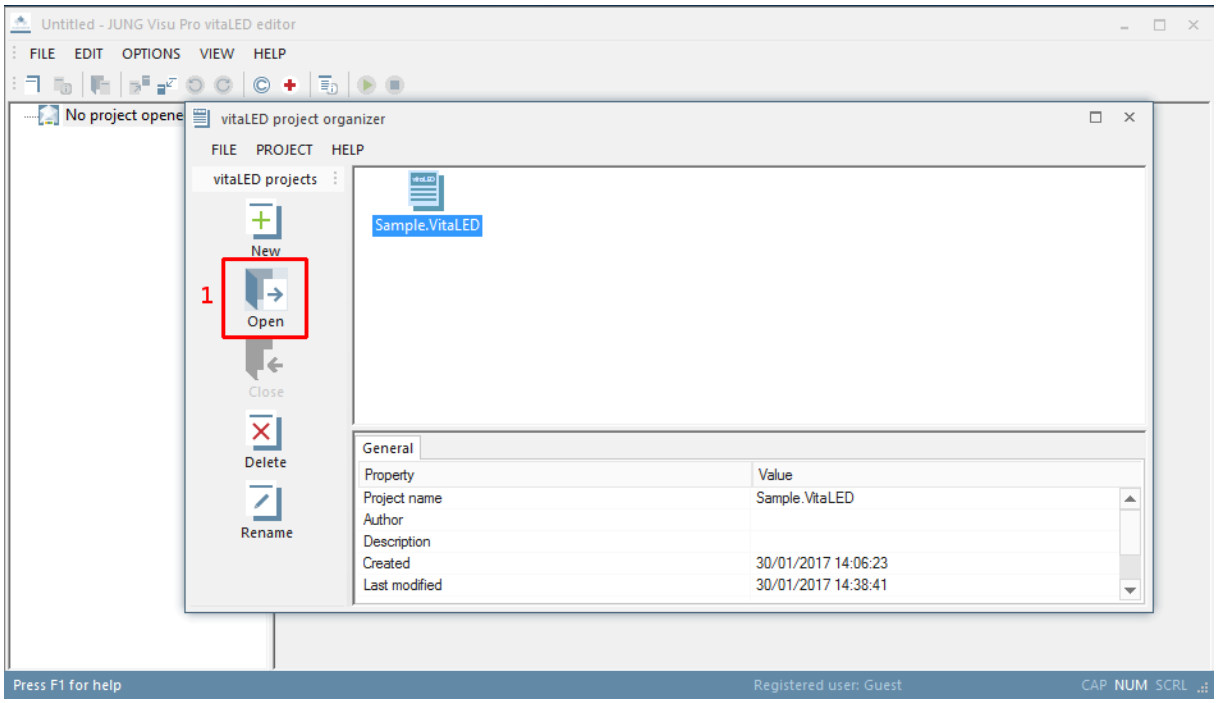
To rename a project, select the desired project and click the button ***Rename*** in the sidebar. The dialog ***Rename project*** opens.



Enter in the name field the new project name and click the button ***OK***. The dialog is closed and the new project name is adopted. The extension ***.VITALED*** is automatically added to the project name.

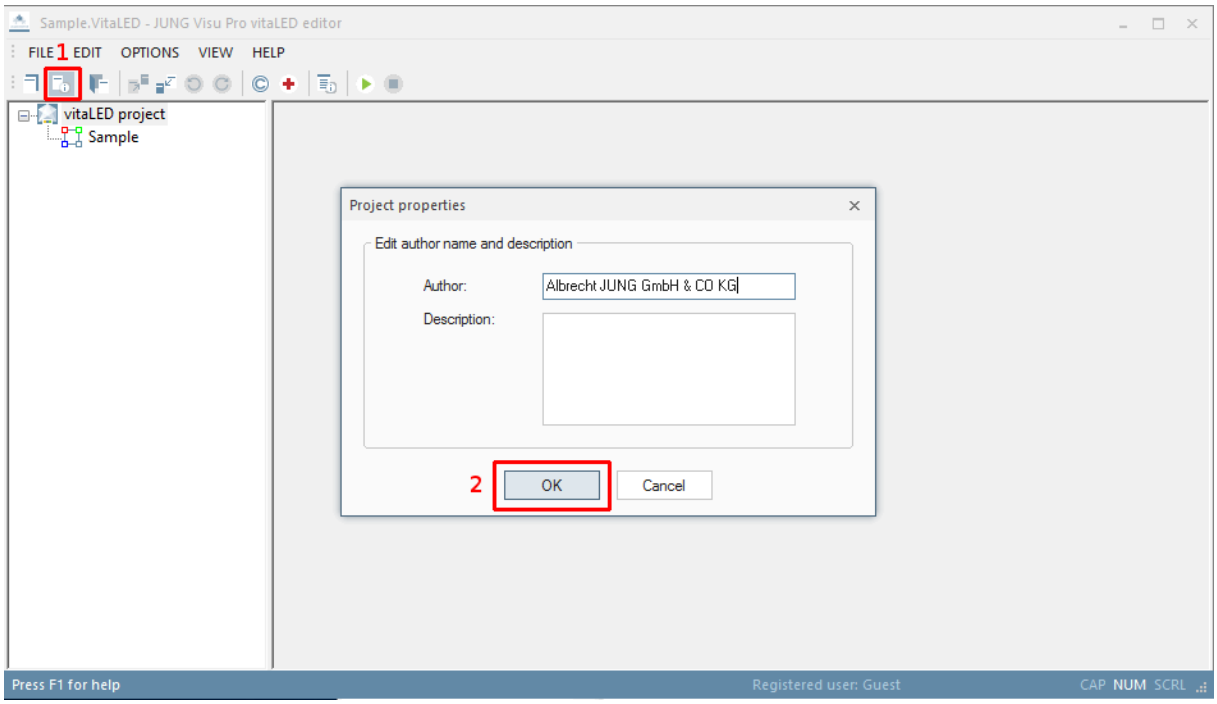
4.1.4 Open vitaLED Project

To load a vitaLED project into the vitaLED editor, select the desired project and click the button ***Open*** in the sidebar.



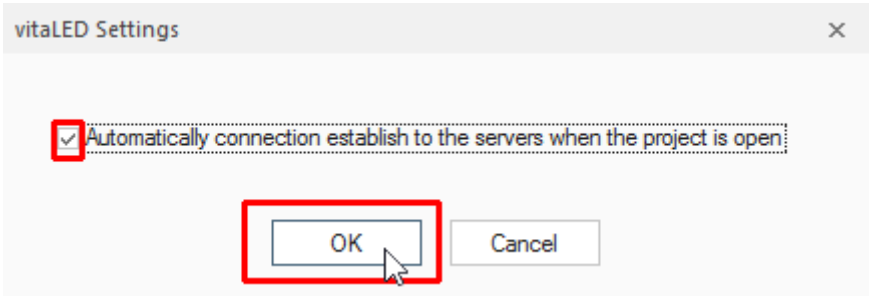
4.2 Edit Project Properties

Choose in the vitaLED editor the menu item ***File*** and click ***Project properties*** or click the symbol in the toolbar to open the dialog ***Project properties***.



Edit the specifications *author* and *description* and click the button ***OK***.

4.3 vitaLED Settings

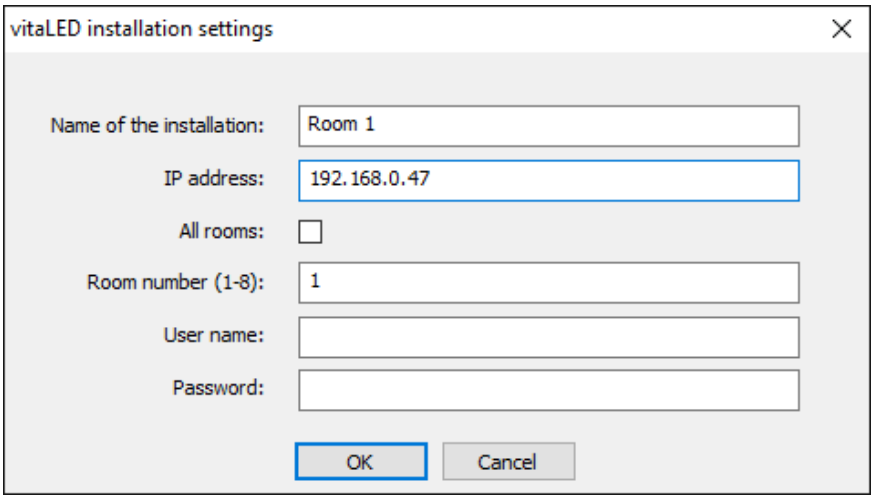


Automatically connection establish to the servers when the project is open:

Click ***vitaLED settings..*** under ***Options*** *to open the settings dialog.*
Check this box to connect automatically to the servers when starting the vitaLED project. The advantage of this setting is that you do not have to establish the connection manually.

4.4 Installation Settings

Click ***Create installation*** in the vitaLED editor under ***Edit*** to open the dialog ***vitaLED installation settings***.



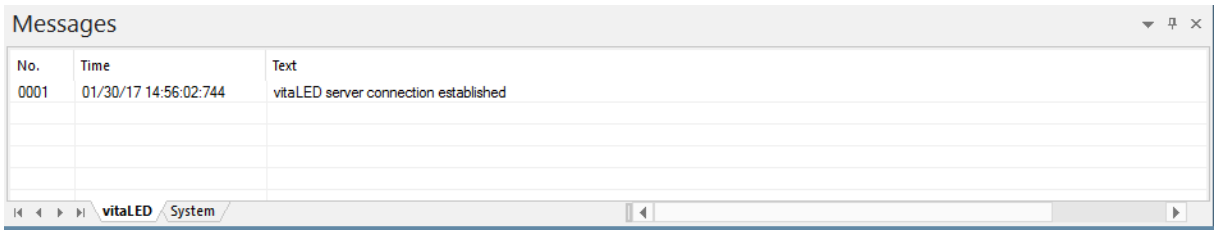
Enter in the field ***name of the installation*** the name and in the field ***IP Address*** the connection to the vitaLED server. Enter the user name and the password as configured on the vitaLED server.
The entered room number must correspond to the address setting of the vitaLED module. Then click the button ***OK***. The dialog is closed and the installation is created.

To edit the installation, click in the vitaLED editor under ***Edit*** on ***Edit installation*** to open the dialog ***vitaLED installation settings*** of the selected installation.

To delete the installation, click in the vitaLED editor under ***Edit*** on ***Delete installation*** to open the dialog ***vitaLED editor*** of the selected installation. Click the button ***OK*** to close the dialog and to delete the installation.

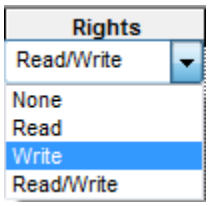
4.5 Connect to vitaLED Servers

To start or to interrupt the connection to the vitaLED servers, click under ***Options*** on ***Start vitaLED*** or ***Stop vitaLED.*** Alternatively, click the symbol or in the toolbar.



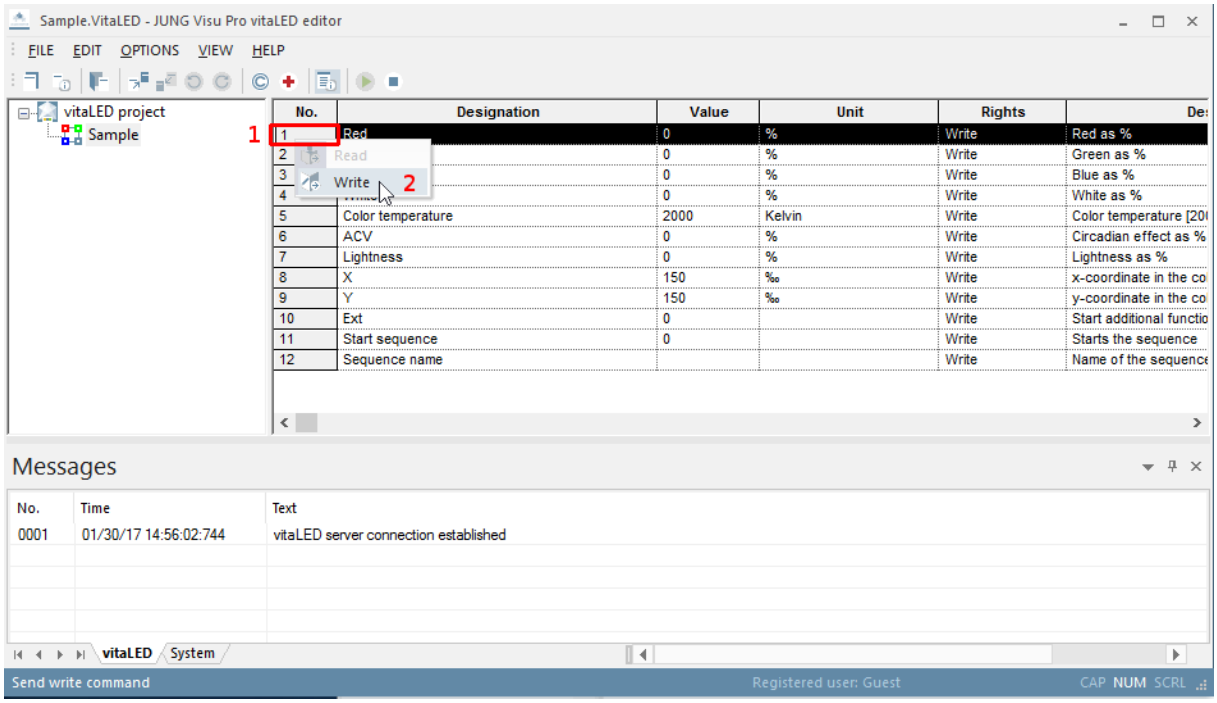
You can see the confirmations or error messages of these operations in the message list.

4.6 Send Telegrams Manually



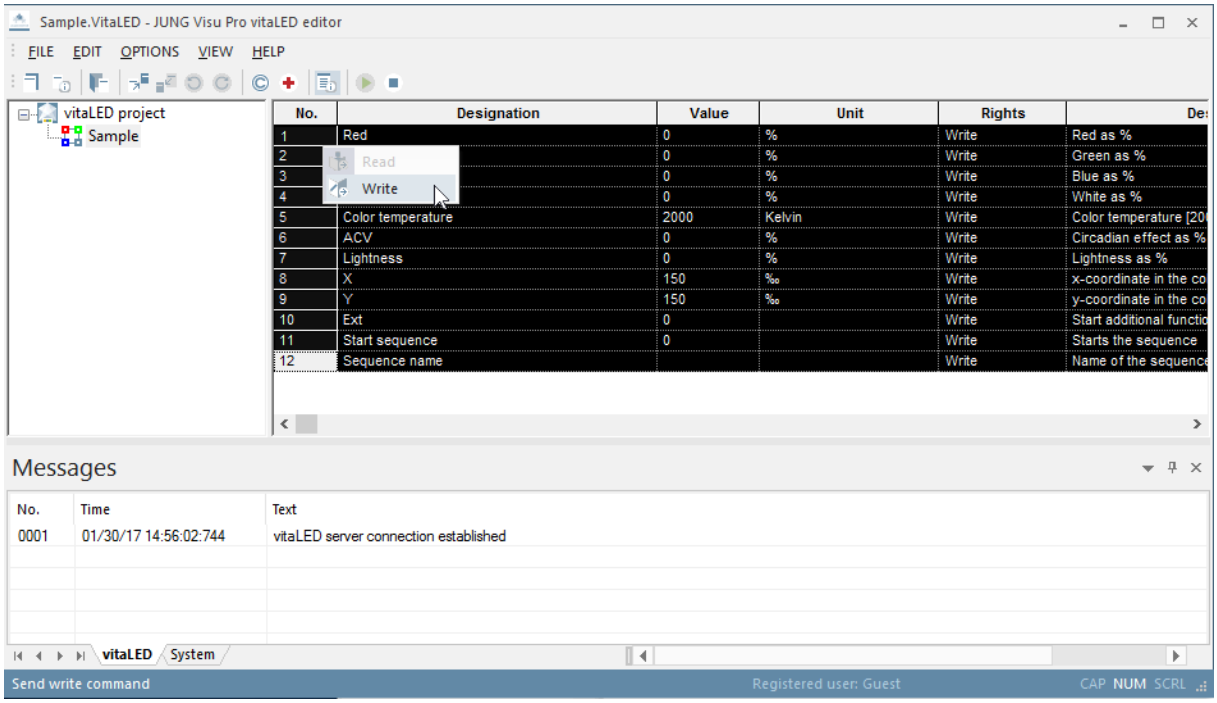
To send a message, first select the access rights required. If you have already done this, enter in the field ***value*** the value to be send.

To send the value, select the entire row with a left click on the ***No.*** colum in the desired row (1). Then right-click and select write command in the context menu (2).



After the command has been performed, the messages are shown in the message list.

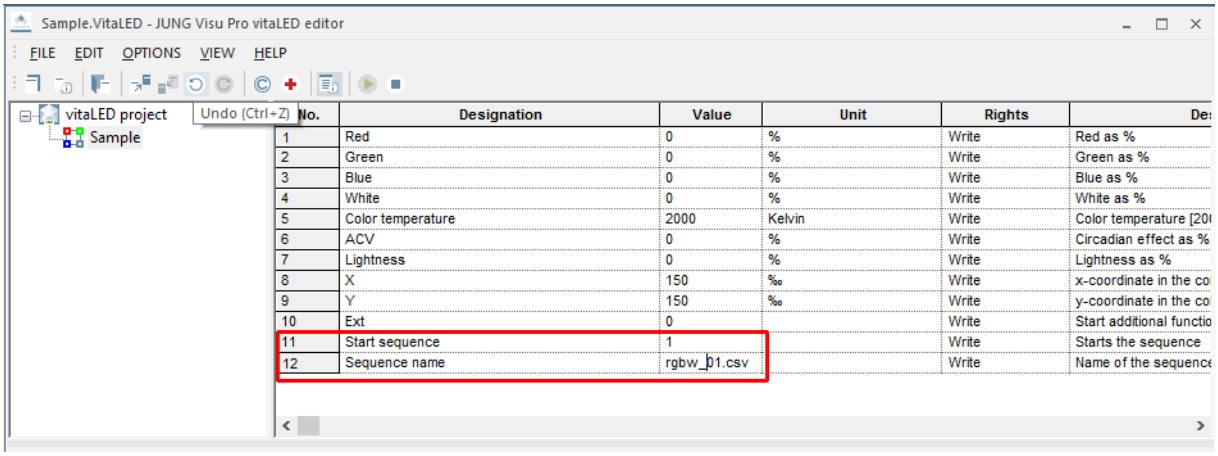
To send several commands at the same time, click and hold the left mouse button and select the commands you want to send. Then perform a right-click and select the write command in the context menu.



4.7 Sequences

The VitaLED-Editor offers the option to send multiple telegrams in a sequence. For that to happen, you have to enter the name of the sequence as the value of **Sequence name** and set the value of **Start sequence** to 1. You find sequences in the form of csv-files in the app directory under **VitaLEDEditor** in the sub folder **Sequences**.

To create your own sequence, you have to create a new csv-file in the **Sequences** folder. To use this sequence, you have to enter the name of the file with the file ending as the value of **Sequence name**.



No.	Designation	Value	Unit	Rights	De
1	Red	0	%	Write	Red as %
2	Green	0	%	Write	Green as %
3	Blue	0	%	Write	Blue as %
4	White	0	%	Write	White as %
5	Color temperature	2000	Kelvin	Write	Color temperature [20
6	ACV	0	%	Write	Circadian effect as %
7	Lightness	0	%	Write	Lightness as %
8	X	150	‰	Write	x-coordinate in the co
9	Y	150	‰	Write	y-coordinate in the co
10	Ext	0		Write	Start additional functio
11	Start sequence	1		Write	Starts the sequence
12	Sequence name	rgbw_01.csv		Write	Name of the sequence

The csv-file contains one command per row. Each value is separated by a '**,**'. A row is constructed as follows:
Red value, green value, blue value, white value, color temperature, ACV, lightness, X, Y, time to wait in seconds, description of the command.

Example:

```
0,0,100,100,,,,,,2,Blue
0,100,0,100,,,,,,2,Green
100,0,0,100,,,,,,2,Red
```

This sequence will send first red then green then blue in an interval of two seconds. All with a white value of 100. Values, which should not be changed can be left empty.

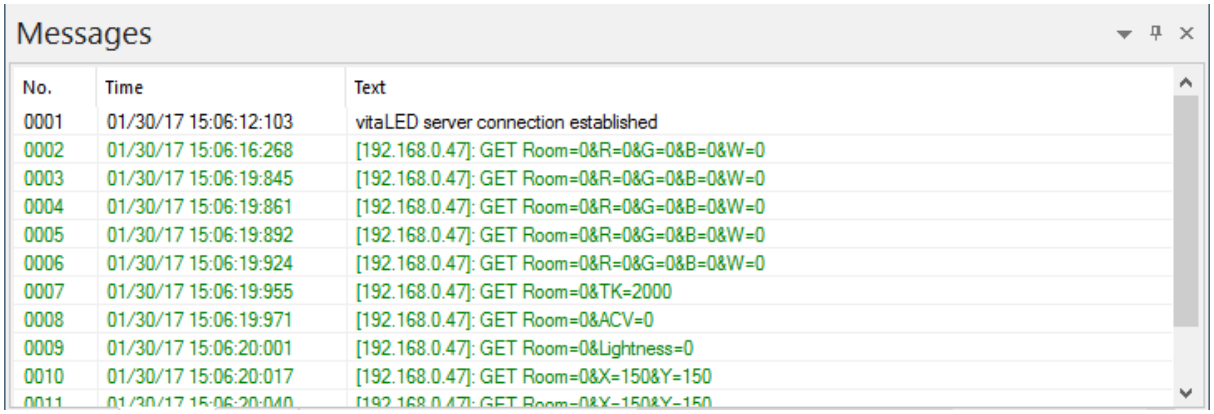
The commands will be called after another as long as the sequence is active. If the end of the file is reached, it will start from the beginning.

4.8 Message List

The message list shows all sent messages (HTTP-Requests).

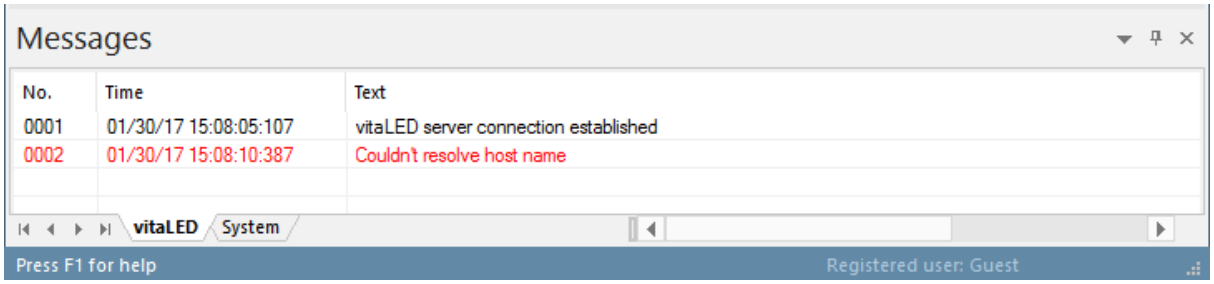
To toggle the visibility of the message list, choose the menu item ***View*** and click ***Messages***. Alternatively, click the symbol in the toolbar.

Choose the menu item ***View*** and click ***Show last message*** to specify that the message list scrolls automatically to the last message.



Meaning of the colors in the message list:

Green (sent messages): As shown in the figure above, the commands were transmitted via messages to the vitaLED server.

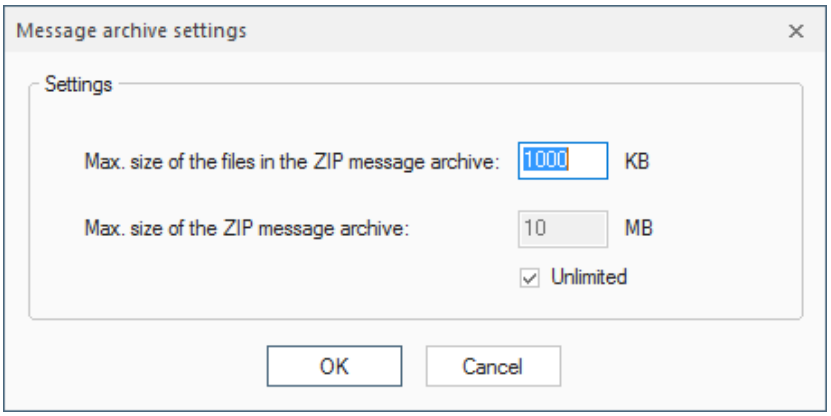


Red: Shows **Status-** or **error messages**.

4.9 Logging vitaLED Messages into Archives

Messages displayed in the message list can be automatically saved in a text file. Choose the menu item ***Options*** and click ***Save vitaLED telegrams*** to activate or deactivate the automatic logging.

To configure the settings for the message archive, choose the menu item ***Options*** and click ***vitaLED archive settings***. The dialog ***Message archive settings*** opens.



When the telegram log file reaches a size of about 1 Megabyte, it will be closed and saved into the zip archive **VitaLEDMessages.zip:**

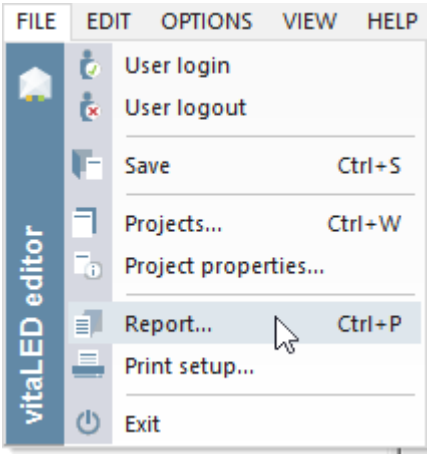
(...\stations\local\projects\<projectname>.VITALED\VitaLEDMessages.zip).

A new log file will be created. The log file will also be closed and saved in the zip archive when the vitaLED connection has been disconnected.

5 Reports

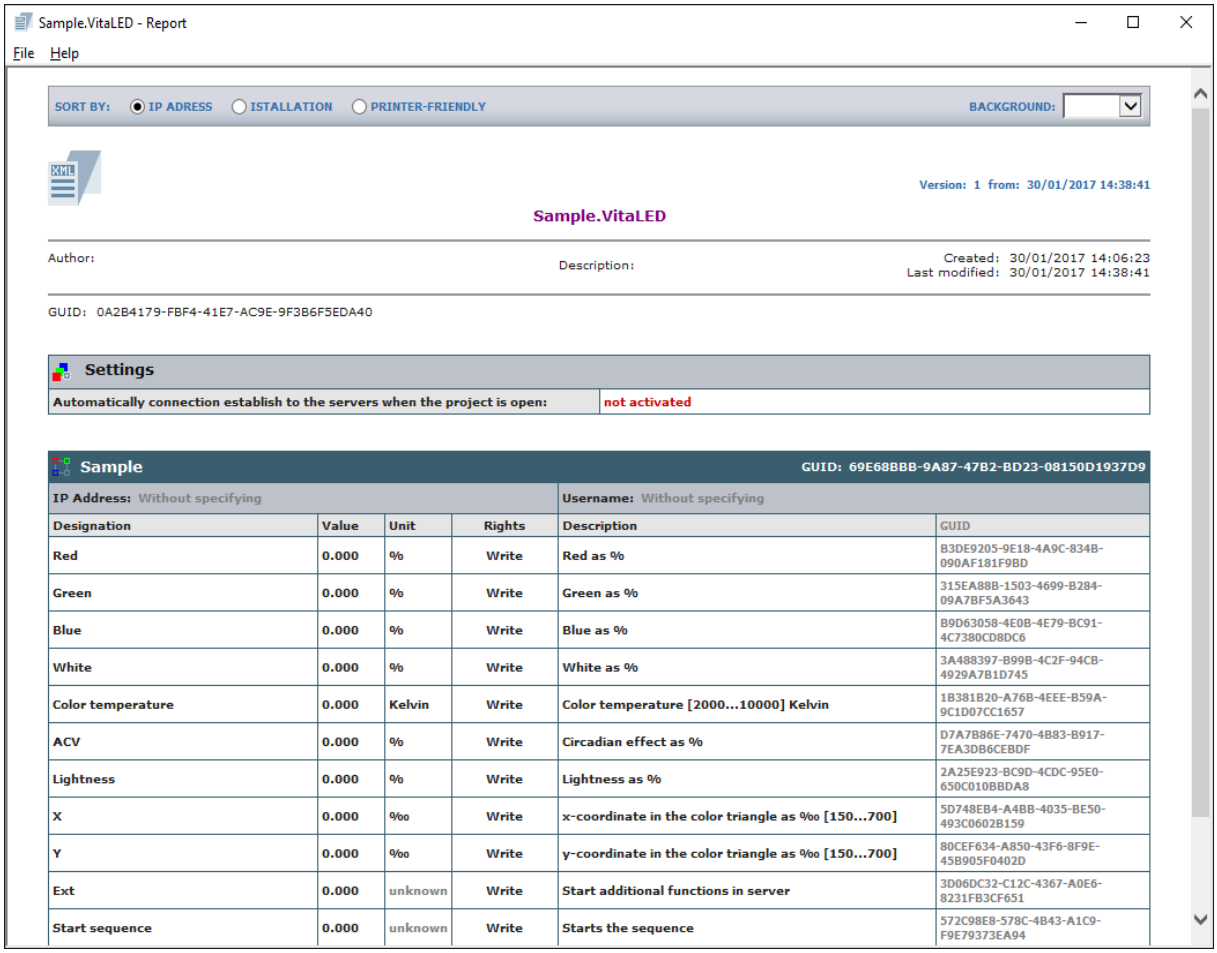
- A report can be created which gives information about the vitaLED installations contained in the project and their properties.
- Reports can be printed out (with print preview).

5.1 Create Report



To create a report for the current project, choose the menu item ***File*** and click ***Report***.

In the menu at the top the report may be sorted by either IP addresses or installations.



Sample.VitaLED - Report

File Help

SORT BY: IP ADRESS ISTALLATION PRINTER-FRIENDLY BACKGROUND:

Version: 1 from: 30/01/2017 14:38:41

Sample.VitaLED

Author: Description: Created: 30/01/2017 14:06:23 Last modified: 30/01/2017 14:38:41

GUID: 0A2B4179-FBF4-41E7-AC9E-9F3B6F5EDA40

Settings

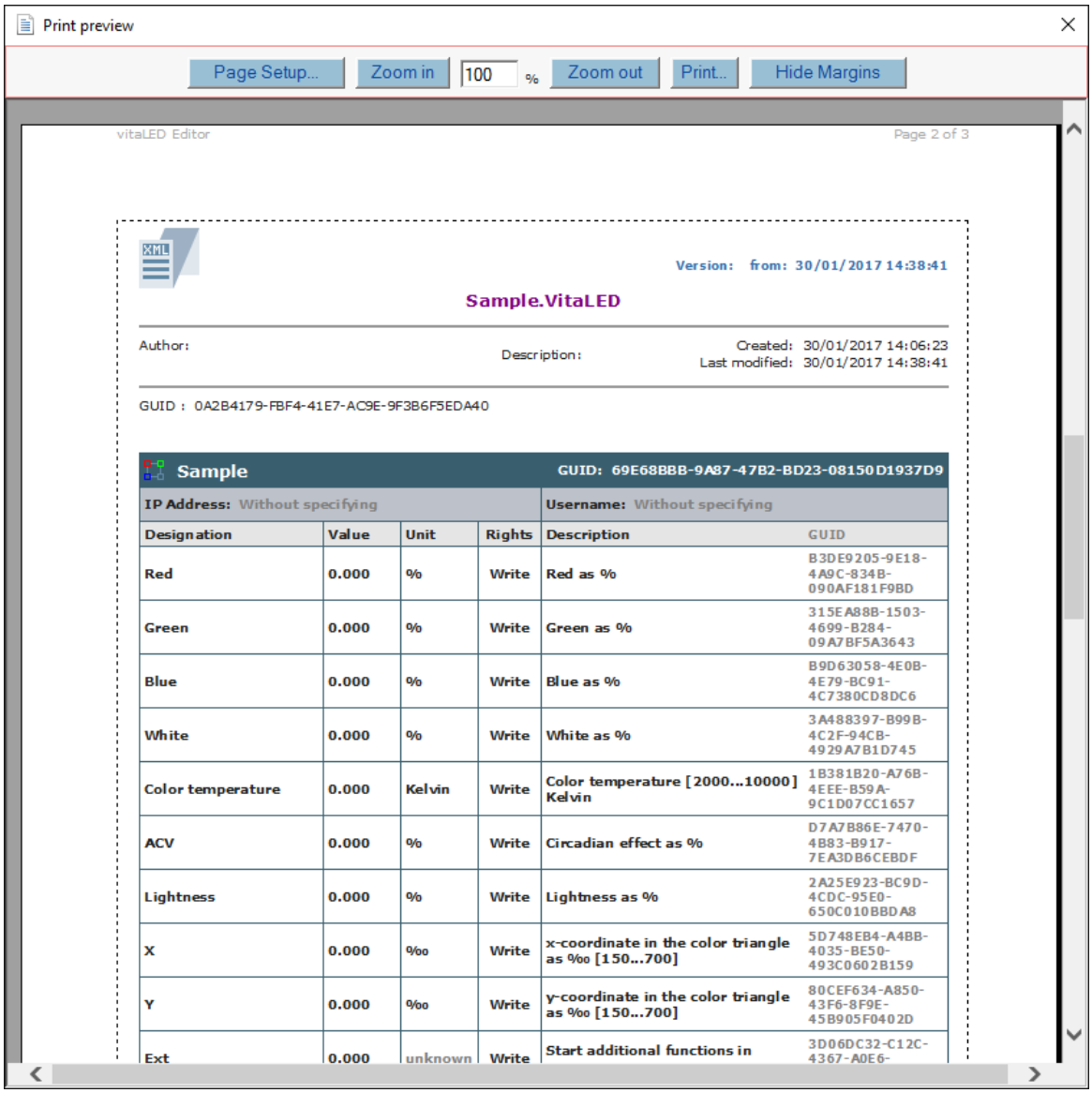
Automatically connection establish to the servers when the project is open:	not activated

Sample GUID: 69E68BBB-9A87-47B2-BD23-08150D1937D9

IP Address: Without specifying Username: Without specifying

Designation	Value	Unit	Rights	Description	GUID
Red	0.000	%	Write	Red as %	B3DE9205-9E18-4A9C-834B-090AF181F9BD
Green	0.000	%	Write	Green as %	315EA88B-1503-4699-B284-09A7BF5A3643
Blue	0.000	%	Write	Blue as %	B9D63058-4E0B-4E79-BC91-4C7380CD8DC6
White	0.000	%	Write	White as %	3A488397-B99B-4C2F-94CB-4929A7B1D745
Color temperature	0.000	Kelvin	Write	Color temperature [2000...10000] Kelvin	1B381B20-A76B-4EEE-B59A-9C1D07CC1657
ACV	0.000	%	Write	Circadian effect as %	D7A7B86E-7470-4B83-B917-7EA3DB6CEBDF
Lightness	0.000	%	Write	Lightness as %	2A25E923-BC9D-4CDC-95E0-650C010BBDA8
X	0.000	‰	Write	x-coordinate in the color triangle as ‰ [150...700]	5D748EB4-A4BB-4035-BE50-493C0602B159
Y	0.000	‰	Write	y-coordinate in the color triangle as ‰ [150...700]	80CEF634-A850-43F6-8F9E-45B905F0402D
Ext	0.000	unknown	Write	Start additional functions in server	3D06DC32-C12C-4367-A0E6-8231FB3CF651
Start sequence	0.000	unknown	Write	Starts the sequence	572C98E8-578C-4B43-A1C9-F9E79373EA94

5.2 Print Preview



The print preview may be scaled.

Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.

Enlarges the print preview.

Reduces the print preview.

Button to hide the margin markers.

Button to display the margin markers.

With these , markers the user has visual control over the margin settings.

JUNG Visu Pro
OPC Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

2.1 First Steps....4

3 USER INTERFACE....8

3.1 Menu....8
3.2 Toolbar....10

4 FUNCTIONS....11

4.1 Project Organizer....11
4.1.1 Create Project....11
4.1.2 Open Project....12
4.1.3 Rename Project....13
4.2 OPC Group Management....14
4.2.1 Create OPC Group....14
4.2.2 Delete OPC Group....15
4.2.3 Edit OPC Group....16
4.3 General Settings....17
4.4 Assign OPC Group....18
4.5 Update OPC Project....21
4.6 OPC Server....22
4.6.1 Server Status....22
4.6.2 Start and Stop OPC Server....22
4.7 Edit Project Properties....24
4.8 Manually Send Telegrams....25

5 REPORTS....26

5.1 Create Report....26
5.2 Print Preview....27

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 Bit per Pixel.
Interfaces	Interface for the connection to the process.

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

In order to clearly distinguish between the different applications, different colors were used. The icon of the **OPC editor** contains a blue colored bar .

The **OPC editor** is part of the visualization system. It is used to connect to the OPC server and helps to put OPC into operation with the **visualization** system.

After the installation of the **visualization system** a program group is created in the Start Menu. The default settings can be adopted or manually changed during installation. You may start the control panel application of the **visualization system** from the WINDOWS program manager. To start an application, select the list entry ***Function Modules*** in the sidebar of the control panel and double-click the icon of the application you wish to start.

2.1 First Steps

After starting the OPC editor, execute the following actions:

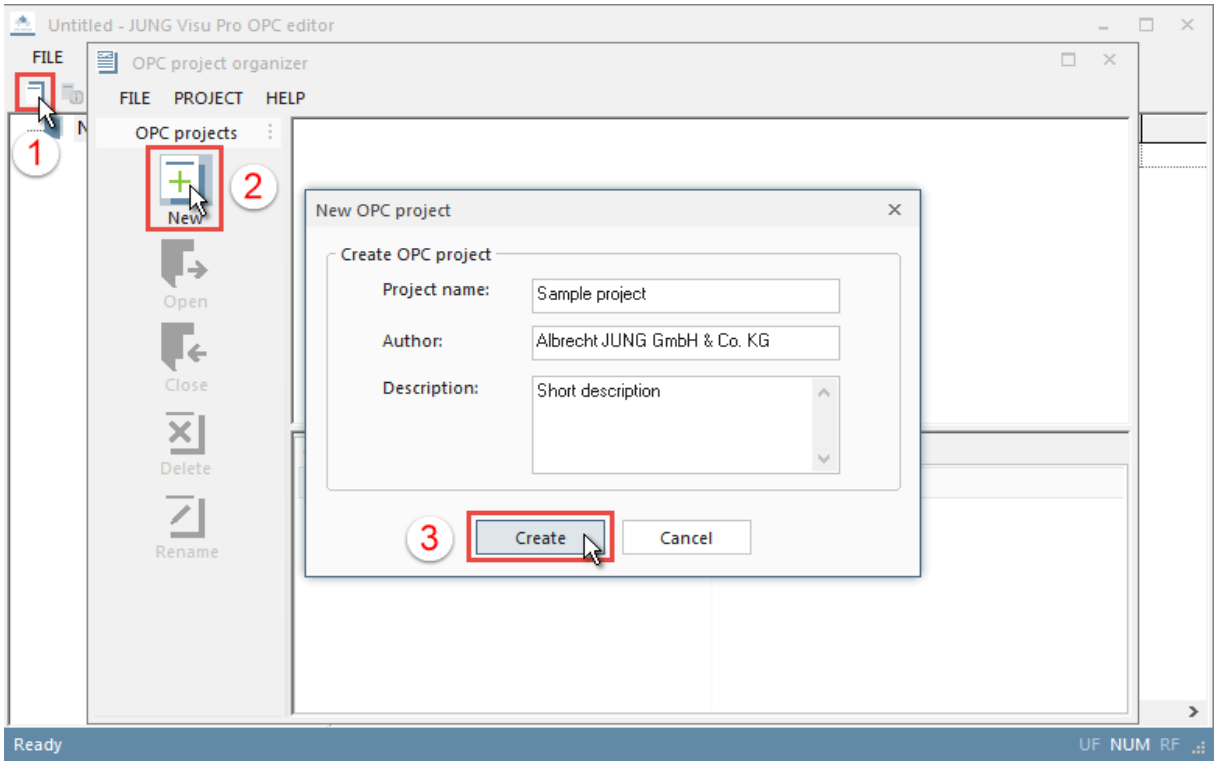
1. Create an OPC project.
2. Select an OPC server.
3. Read the OPC project from the selected OPC server.

1. Create OPC Project

New projects can be created in the ***Project organizer***. When the OPC editor is started, the project organizer dialog opens automatically. To open the project organizer manually, choose in the OPC editor the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

In the project organizer you can use the buttons in the sidebar or the entries under the menu item ***Project***.

To create a new project, click the button ***New*** in the sidebar. The dialog ***New OPC project*** opens.



Enter the following information:

Project name: Unique name of the OPC project.

Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or edited later.

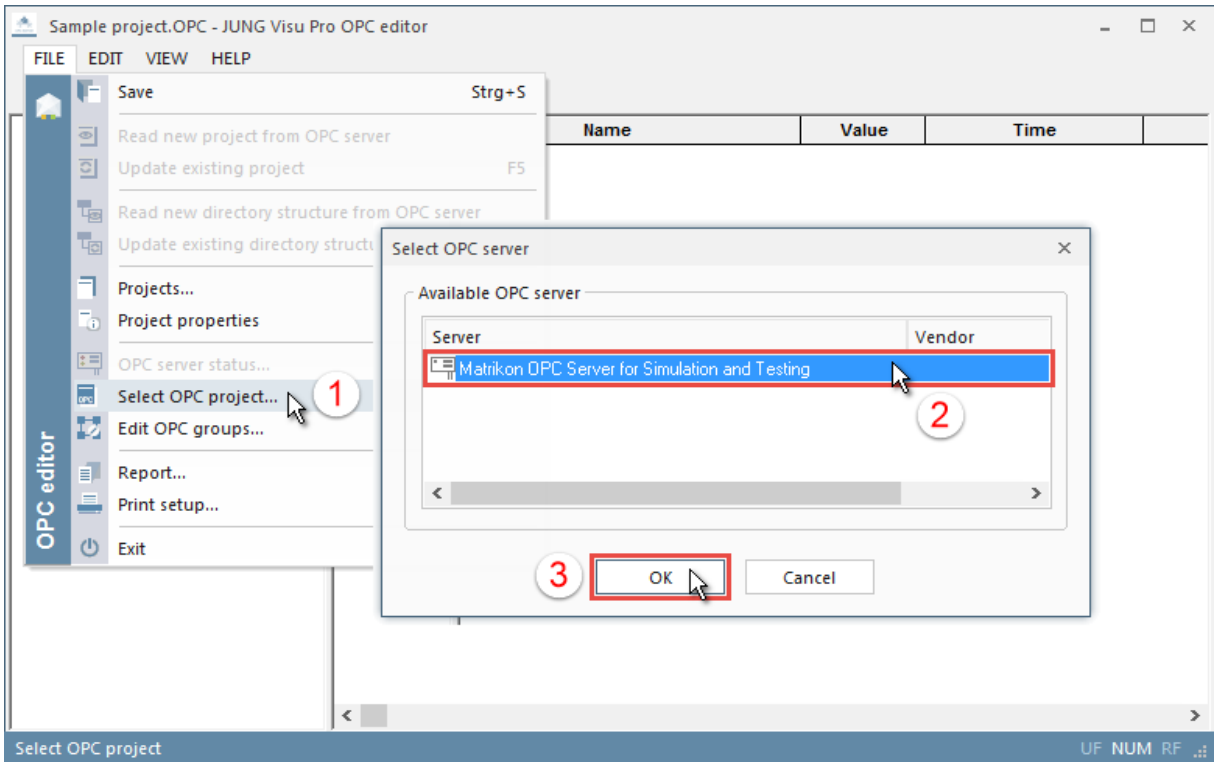
Create: Click the button ***Create*** to complete the creation of the project. It is now available in the project list of the project organizer dialog.

To load the newly created project into the OPC editor,

- double-click the project symbol or
- select the project, choose the menu item ***Project*** and click ***Open*** or
- select the project and click the symbol ***Open*** in the sidebar.

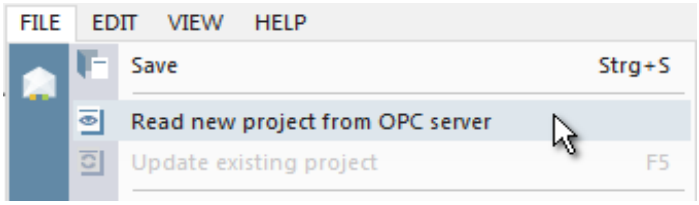
2. Select OPC Project

In the OPC editor choose the menu item ***File*** and click ***Select OPC project***. The dialog ***Select OPC server*** opens.

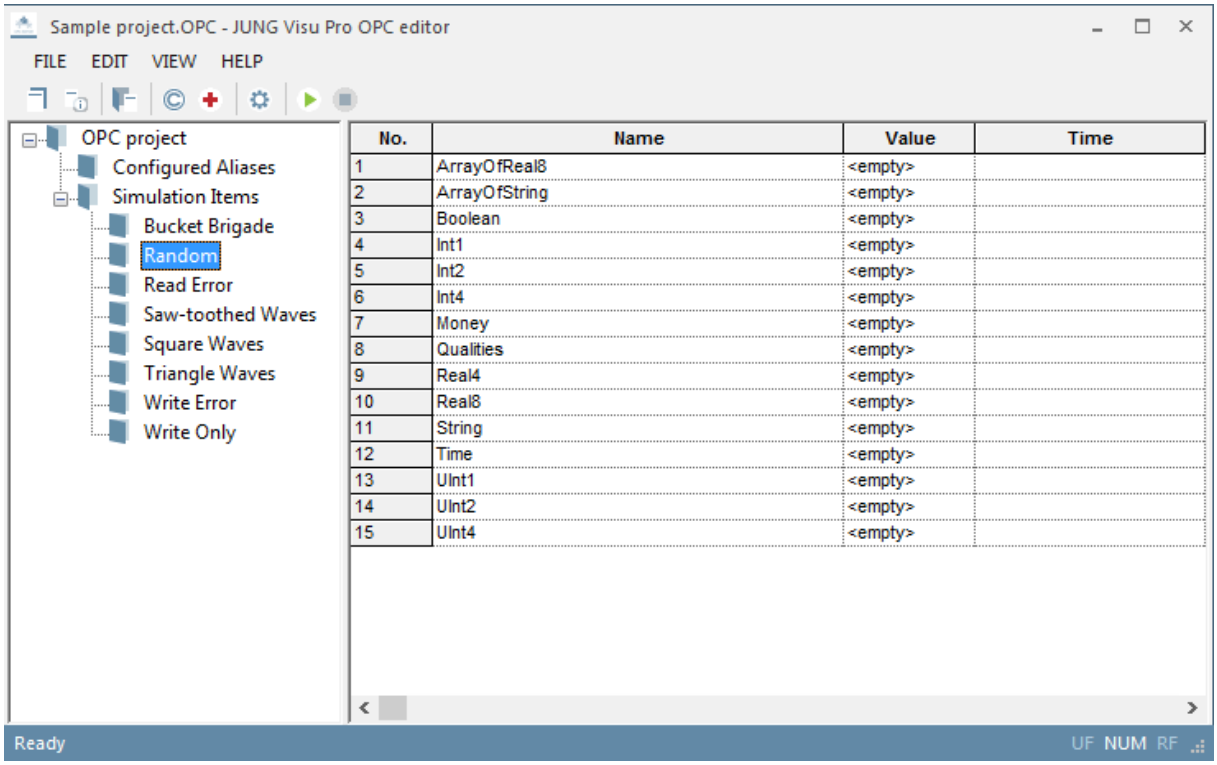


Select an installed OPC server from the list and click the button ***OK***. If the selected server is already used, the corresponding message will be shown.

3. Read OPC Project



After selecting an OPC server, choose the menu item ***File*** and click ***Read new project from OPC server***. The OPC editor then reads the entire project, i.e. the project structure (directory tree) including the data points.

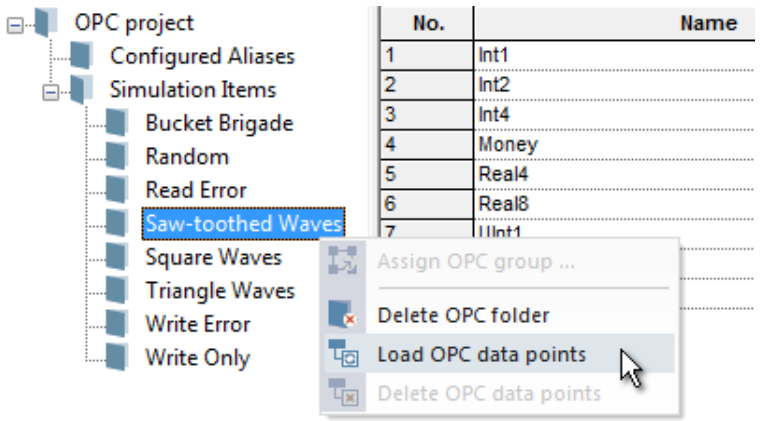


For larger projects it is also possible to import only the parts that are required.

The menu option ***Read new directory structure from OPC server*** under the menu item ***File*** allows to import only the OPC project structure, i.e. the directory tree without data points.

Subsequently you can delete unneeded directories and/or subdirectories from the project structure. Click the right mouse button on the desired directory and select in the context menu the option ***Delete OPC folder***.

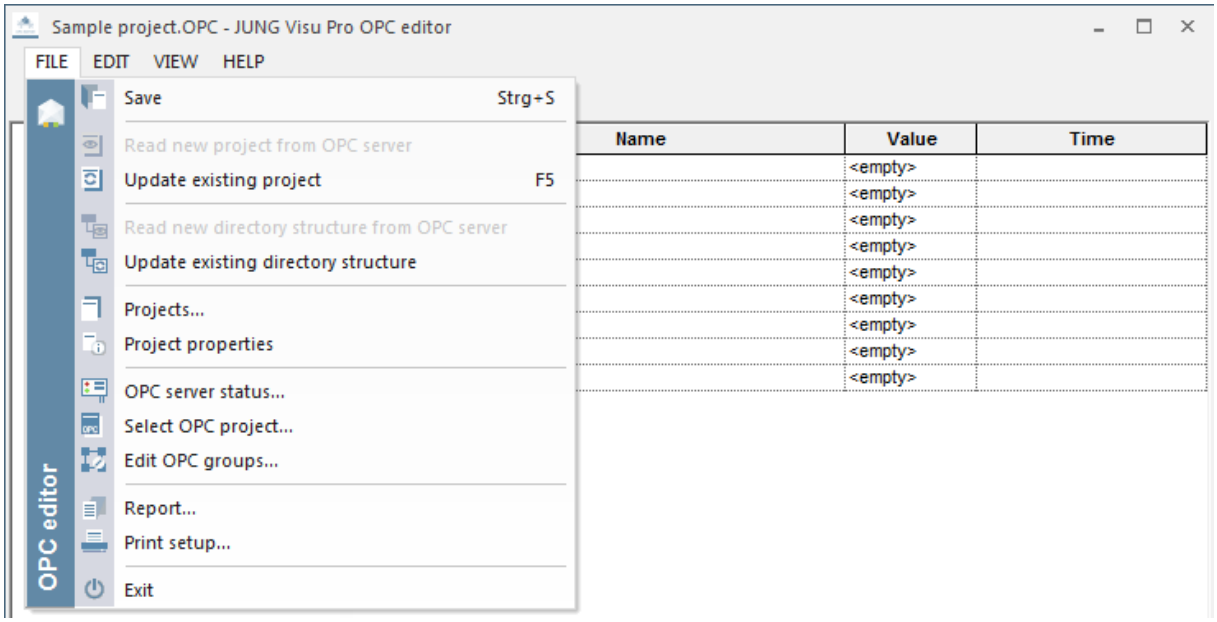
The function ***Load OPC data points*** under the menu item ***Edit*** allows to read the data points of the selected subdirectory from the OPC Server and to update them in the current OPC project.



3 User Interface

3.1 Menu

Menu item *File*



Save: Saves the current project.

Read new project from OPC server: After the selection of an OPC server, this option enables the OPC editor to read the entire project, i.e. the project structure (directory tree) including the data points.

Update existing project: The complete OPC project will be read from the server and updated in the OPC editor (Shortcut F5).

Read new directory structure from OPC server: Only the OPC project structure, i.e. the directory tree without the data points will be read.

Update existing directory structure: Only the existing OPC project structure from the current OPC project will be read.

Projects: Opens the project management dialog.
Project properties: Edit author name and project description.

OPC server status: Opens a dialog displaying OPC server status information.

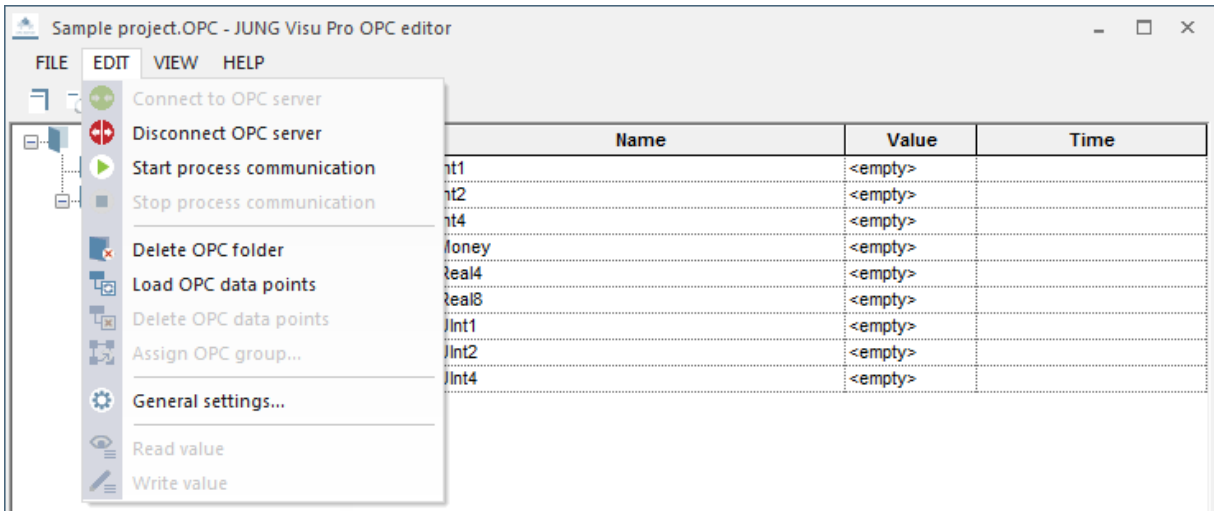
Select OPC project: Opens a dialog to select an installed OPC server from a list.

Edit OPC groups: Opens a dialog to edit the OPC groups.

Report: Generates a new report with print preview for the current project.
Print setup: Selection of printer and setting of printer properties.

Exit: Closes the OPC editor. Shortcut ALT+F4.

Menu item *Edit*



Connect/Disconnect to OPC server: Establishes or interrupts the connection to the OPC server.

Start/Stop process communication: Starts or stops the process communication with the active OPC server.

Delete OPC folder: Deletes the selected folder from the directory tree.

Load OPC data points: Reads the data points of the selected subdirectory from OPC server.

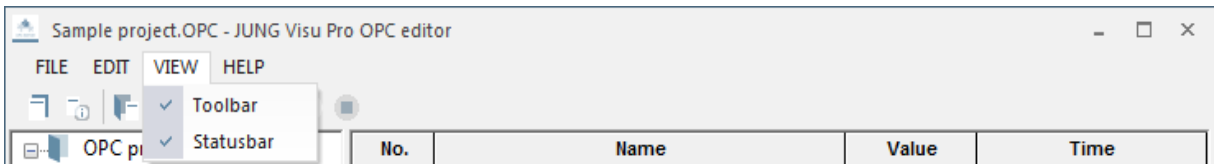
Delete OPC data points: Deletes the selected data points from the selected subdirectory.

Assign OPC group: Opens a dialog to select an OPC group.

General settings: Opens a dialog to configure OPC server settings.

Read and write value: Sends the selected command.

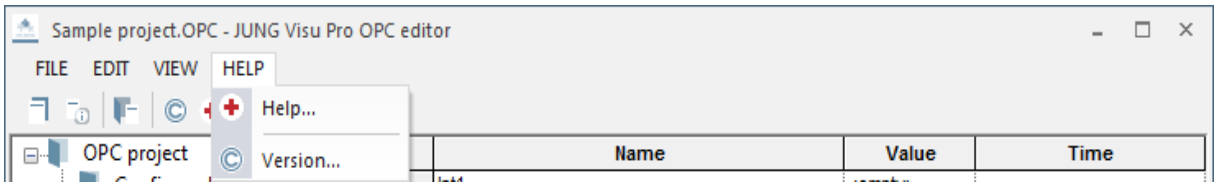
Menu item *View*



Toolbar: Shows or hides general toolbar.

Statusbar: Shows or hides the status bar.

Menu item *Help*



Help: Opens the help file for the program.

Version: Shows version and copyright information.

3.2 Toolbar

Project organizer: Opens a dialog to create and maintain OPC projects.

Project properties: Opens a dialog to edit general project properties.

Save: Saves the current project.

Version: Shows version and copyright information.

Help: Opens the help file of the OPC editor.

General settings: Opens a dialog to edit OPC settings

Start process communication: Starts the process communication with the active OPC server.

Stop process communication: Stops the process communication with the active OPC server.

4 Functions

4.1 Project Organizer

The dialog ***OPC project organizer*** opens automatically when the OPC editor is started. To open the project organizer manually in the OPC editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

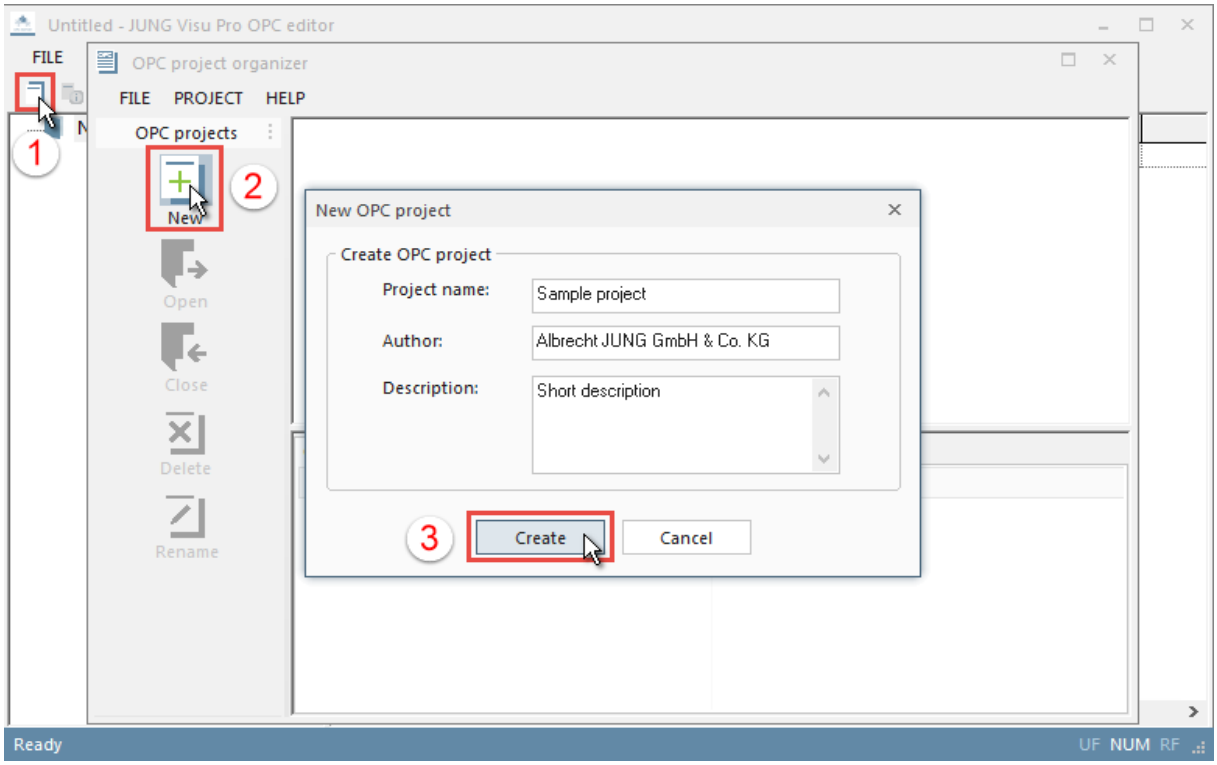
In the project organizer you can create new projects and open, delete or rename existing projects. You can use the buttons in the sidebar of the dialog or the entries under the menu item ***Project***.

The project organizer provides the following options:

- **New**: Create a new OPC project.
- **Open**: Open the selected project.
- **Rename**: Rename the selected project.
- **Close**: Close the currently opened project.
- **Delete**: Delete the selected project.

4.1.1 Create Project

To create a new OPC project, click the button ***New*** in the sidebar. The dialog ***New OPC project*** opens.



Enter the following information:

Project name: Unique name of the OPC project.

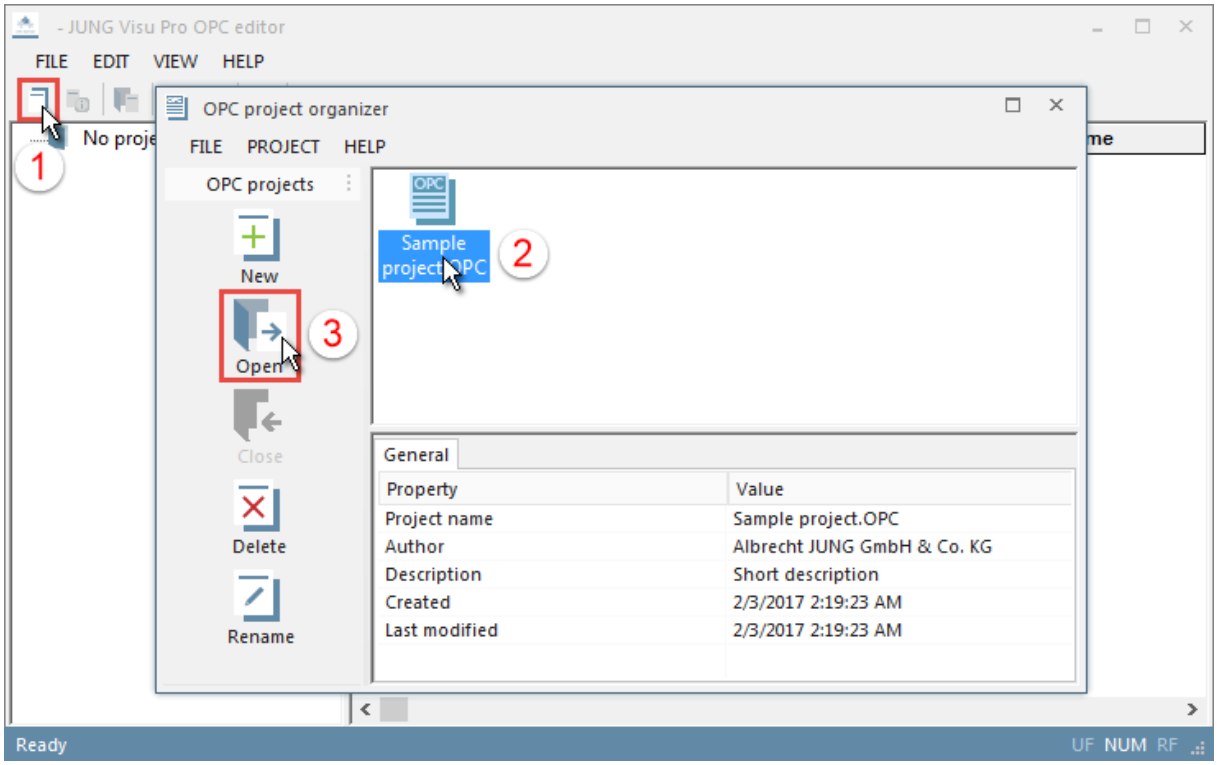
Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or changed later.

Create: Click the button ***Create*** to complete the creation of the project. The peoject is now available in the project list of the project organizer dialog.

4.1.2 Open Project

To load the newly created project into the OPC editor,

- double-click the project symbol in the project list or
- select the project, choose the menu item ***Project*** and click ***Open*** or
- select the project and click the button ***Open*** in the sidebar.

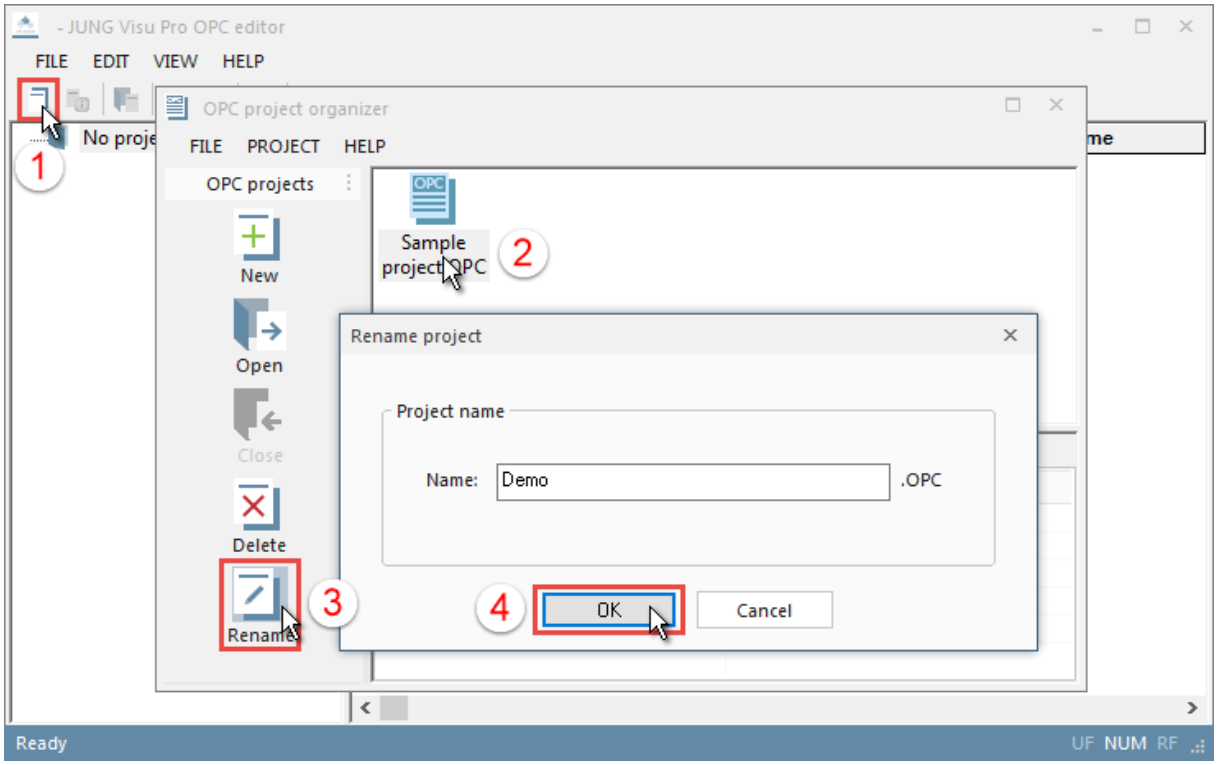


4.1.3 Rename Project

To rename a project, select the project to be renamed,

- choose the menu item ***Project*** and click ***Rename*** or
- click the button ***Rename*** in the sidebar.

The dialog ***Rename project*** opens.



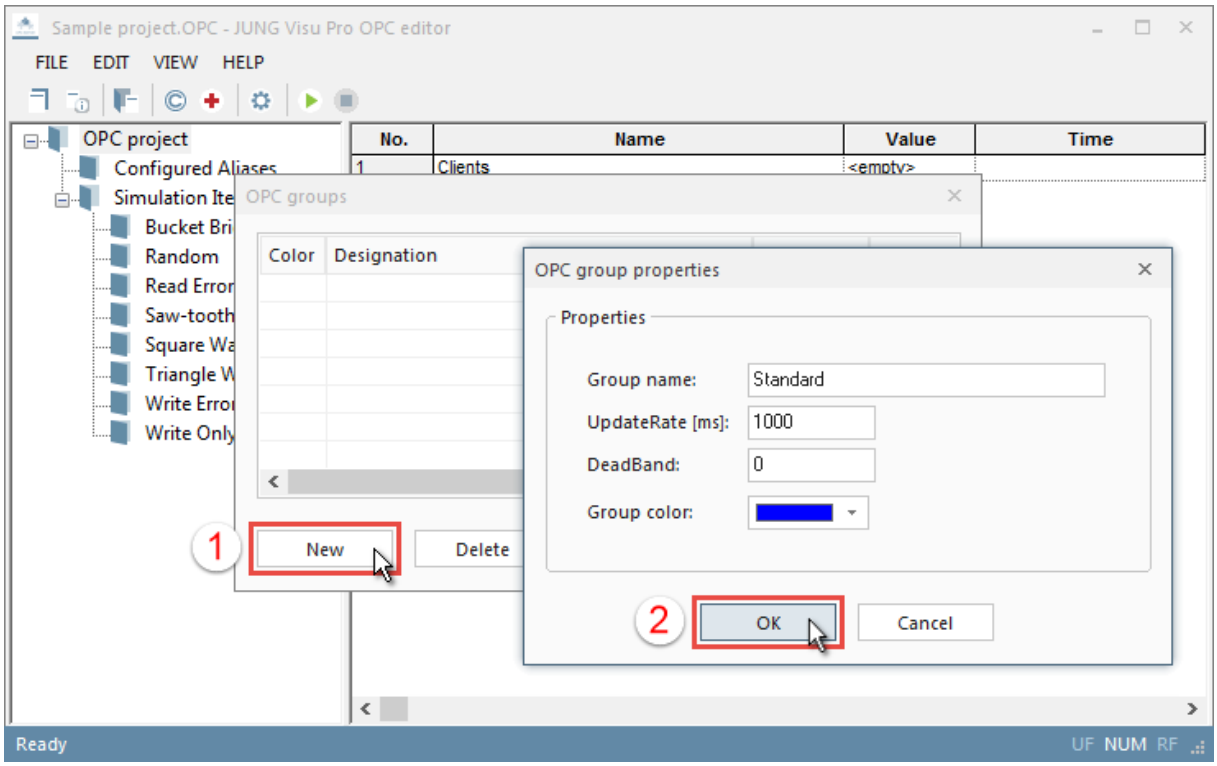
Enter a new name for the project and click the button ***OK*** to close the dialog and to submit the changes. The extension ***.OPC*** is automatically added to the project name.

4.2 OPC Group Management

4.2.1 Create OPC Group

To create an OPC group, choose the menu item ***File*** and click ***Edit OPC groups***. The dialog ***OPC Groups*** opens.

In the dialog click the button ***New***. The dialog ***OPC group properties*** opens.



Enter the following information:

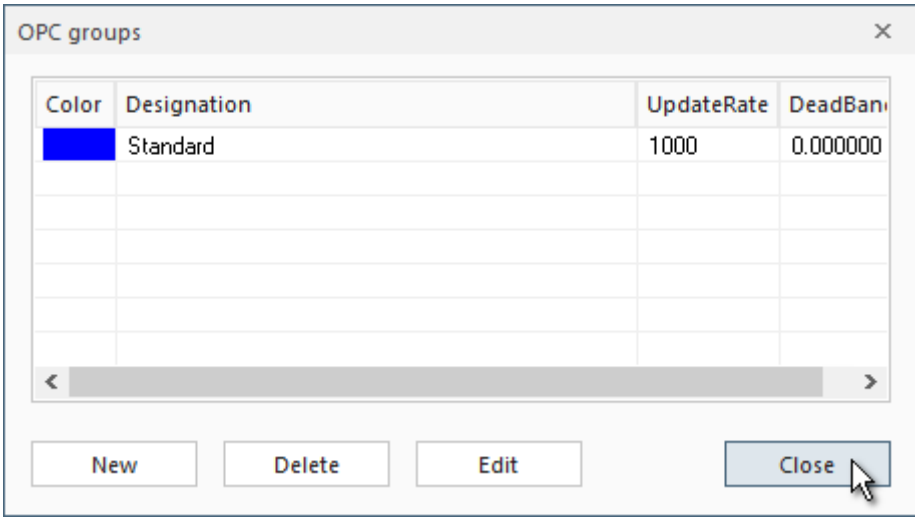
Name: Enter a designation for the group.

UpdateRate [ms]: Enter the update rate in milliseconds.

DeadBand: Enter the tolerance in %.

Color: Select a color for the group.

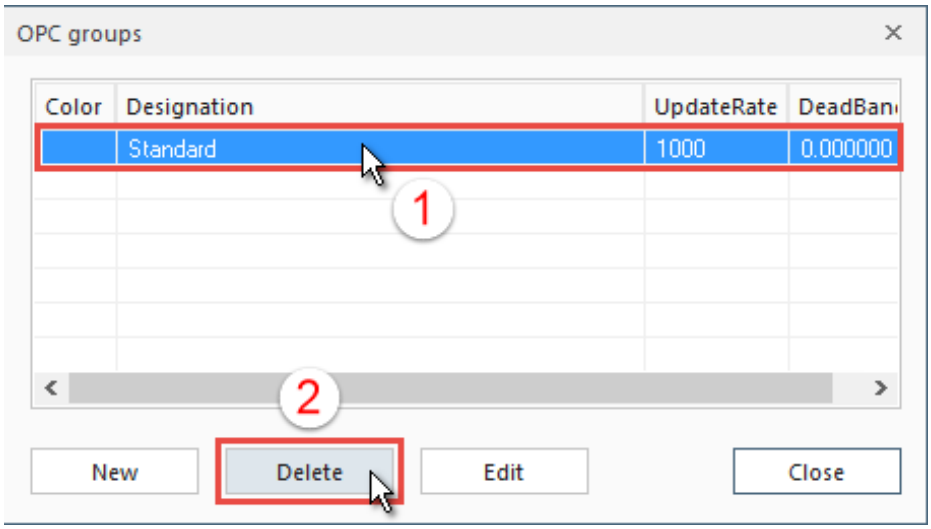
Click the button ***OK*** to submit the entered properties for the group.



Click the button ***Close*** to close the dialog.

4.2.2 Delete OPC Group

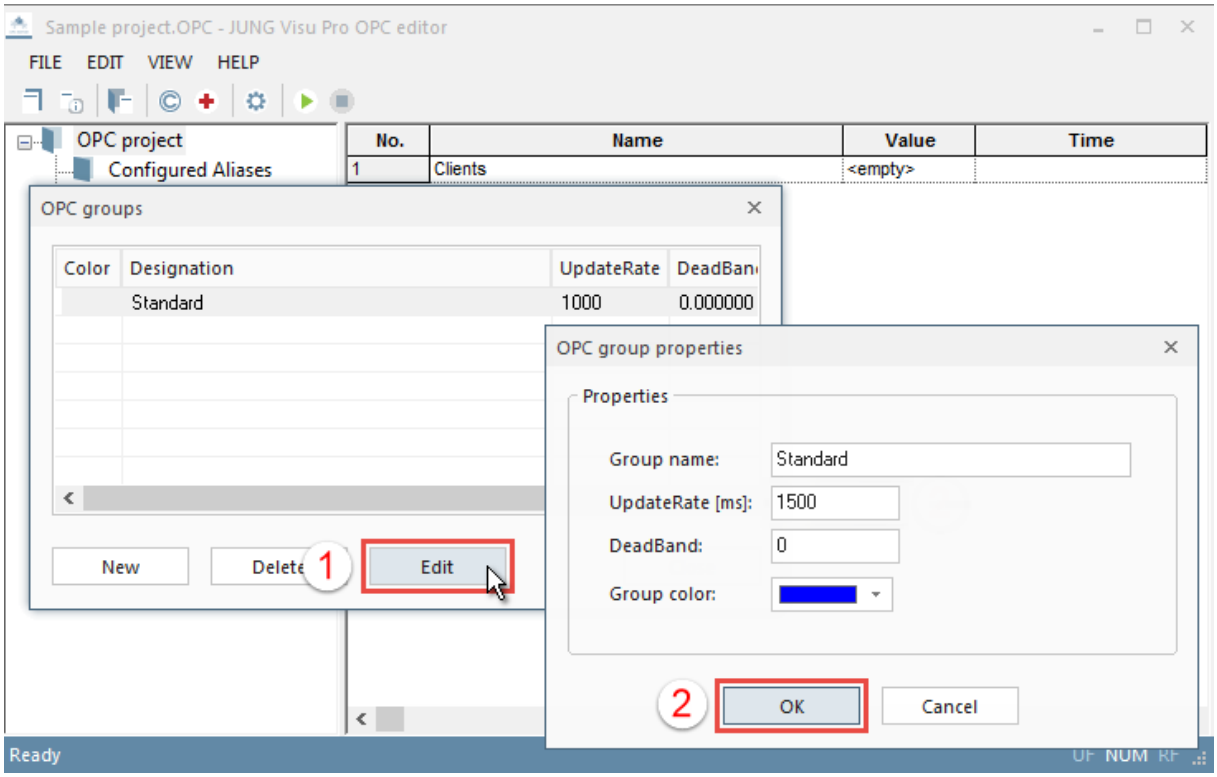
To delete an OPC group, choose the menu item ***File*** and click ***Edit OPC groups***. The dialog ***OPC groups*** opens.



Select the group you want to delete and click the button ***Delete***.

4.2.3 Edit OPC Group

To edit an OPC group, choose the menu item ***File*** and click ***Edit OPC groups***. The dialog ***OPC groups*** opens. Select the group you want to edit and click the button ***Edit***. The dialog ***Group properties*** opens.



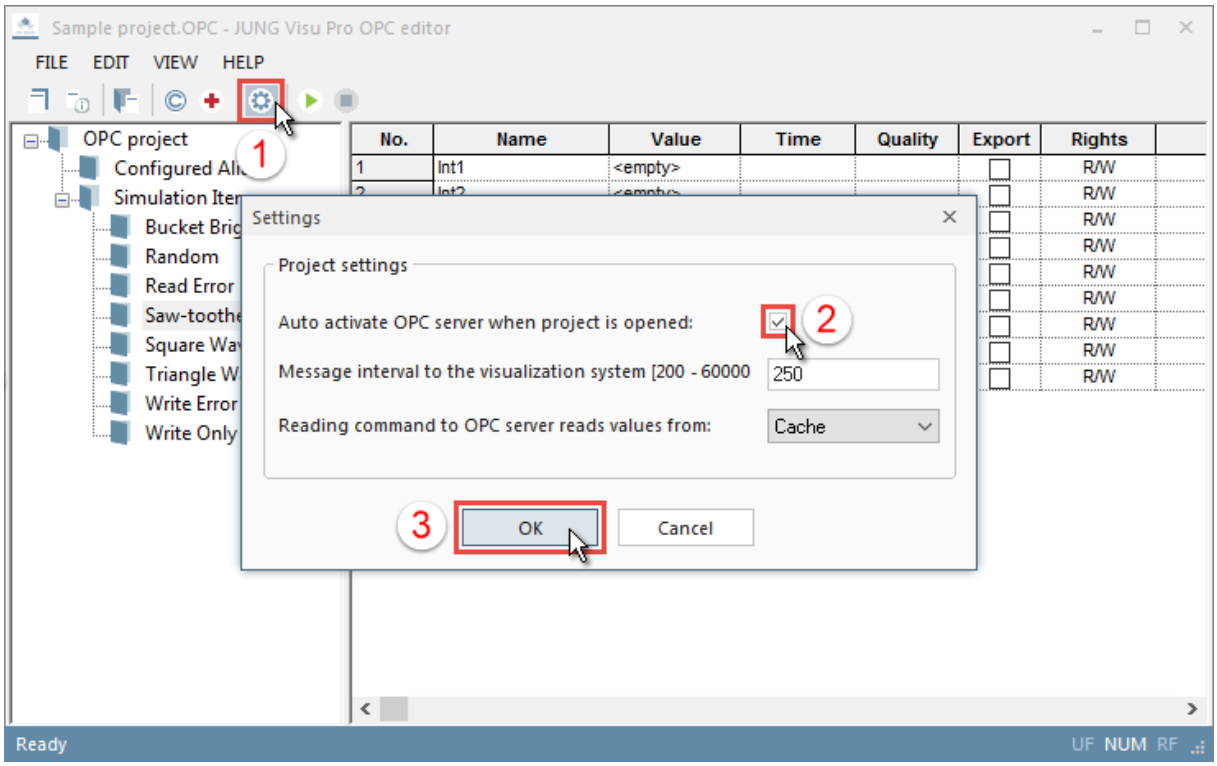
Edit the data and click the button ***OK***.

In the ***OPC groups*** dialog click the button ***Close*** to quit the dialog. The assigned groups of the project will be updated automatically.

4.3 General Settings

Choose the menu item ***Edit*** and click ***General settings*** or click the button in the toolbar.

The dialog ***Settings*** opens.



Enter the following information:

Auto activate OPC server when project is open: After the project is opened, the connection to the selected server will be automatically established.

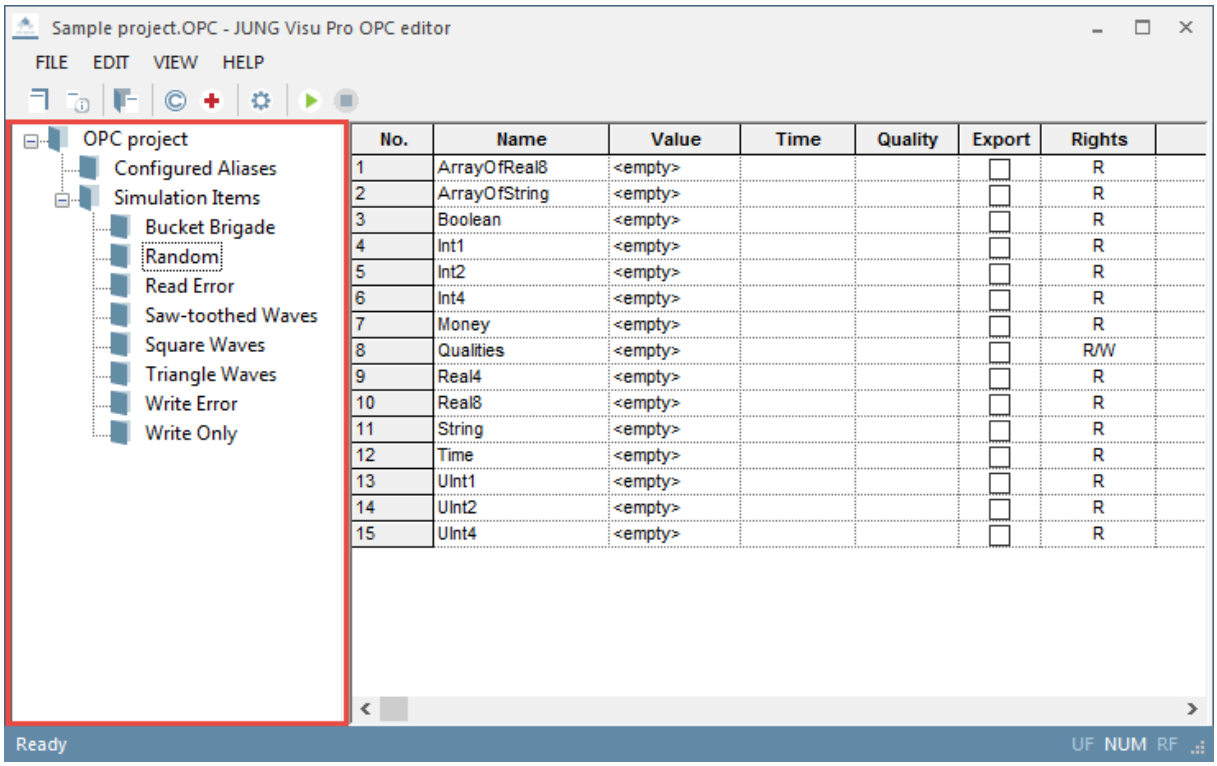
Interval [ms]: Message interval settings to the visualization system in milliseconds.

Reading command to OPC server: Reads the values alternatively from the device or from the cache.

After editing the settings click the button ***OK***.

4.4 Assign OPC Group

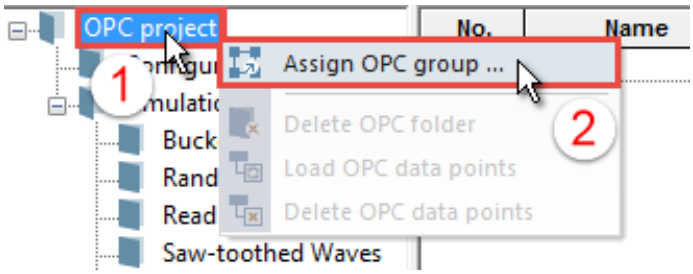
For each group address further specifications have to be made which enable the OPC editor to interpret the data coming from the bus:



Note: *The OPC project is displayed in form of a directory tree in the left window of the OPC editor. The structure of the tree is transferred from the OPC server and therefore cannot be changed in the OPC editor.*

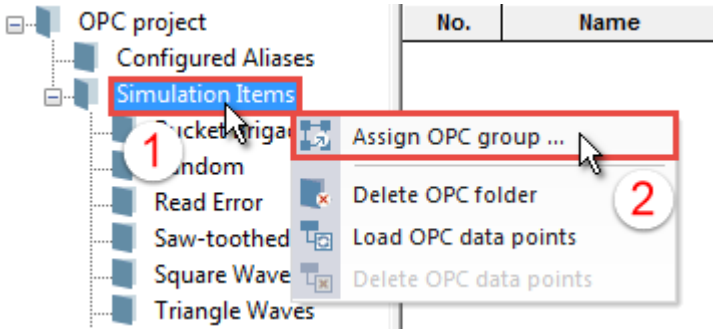
Assign a group to all data points of the OPC project:

First select the OPC project (root) in the directory tree. Click the right mouse button on the selection to open the context menu. In the context menu select the option ***Assign OPC group***. The dialog ***Select group*** opens.



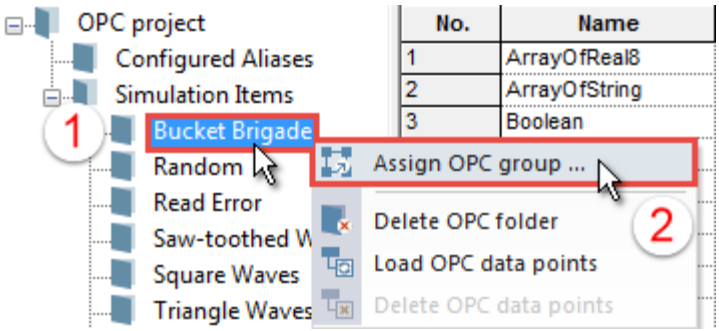
Assign a group to all data points of a directory:

First select the desired directory in the directory tree. Click the right mouse button on the selection to open the context menu. In the context menu select the option ***Assign OPC group***. The dialog ***Select group*** opens.

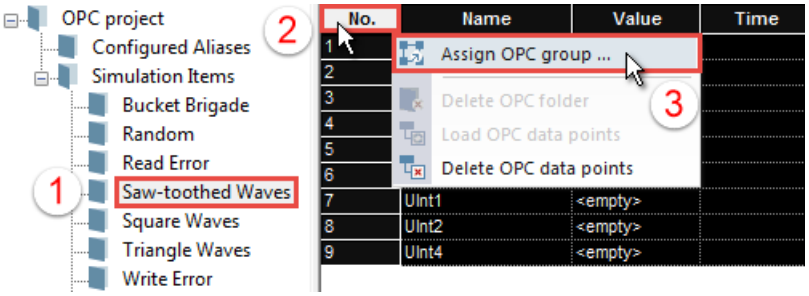


Assign a group to all data points of a subdirectory:

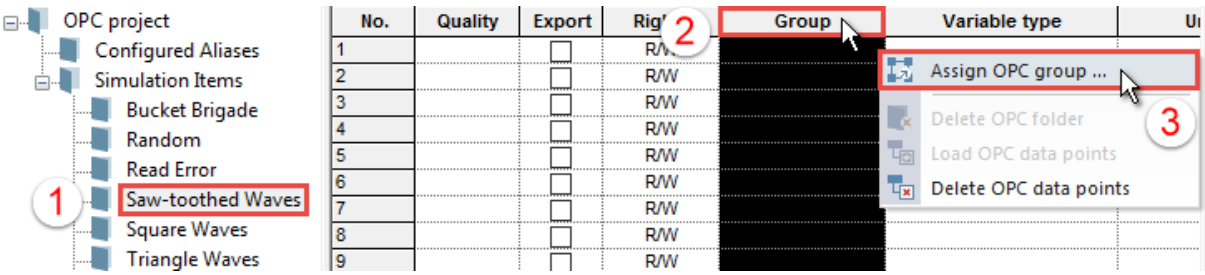
Select the subdirectory in the directory tree.



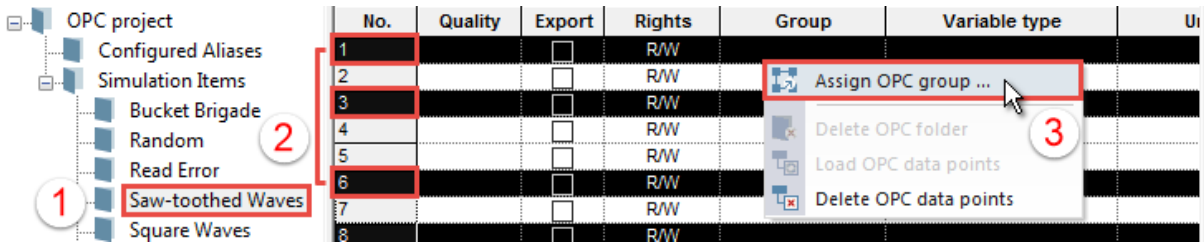
Alternatively: Click the table header ***No.*** to select all data points.



Alternatively: Click the table header ***Group***. Then click the right mouse button and select in the context menu the option ***Assign OPC group***. The dialog ***Select group*** opens.



Assign a group to single data points:



Select single OPC data points:

Select the desired data point by clicking in the column ***No.*** To extend or reduce the selection, click other data points while holding down the Ctrl key.

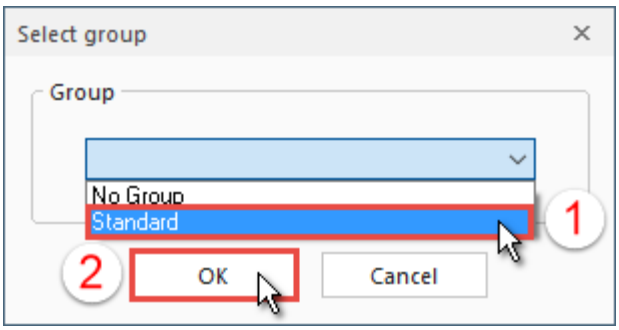
Select a range of OPC data points:

Select the first desired data point in the column ***No.*** (e.g. Number 1), hold down the shift key and select the last desired data point (e.g. Number 9).

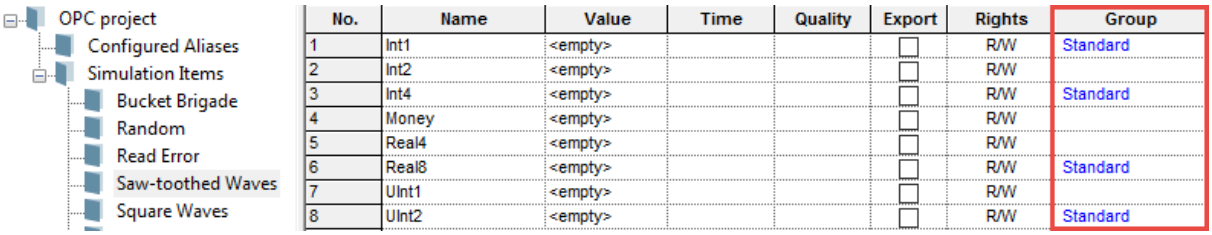
Alternatively: Select the first desired data point, hold down the left mouse button and drag the cursor to the last desired data point.

Then click the right mouse button on the selection to open the context menu. In the context menu select the option ***Assign OPC group***.

The dialog ***Select group*** opens.

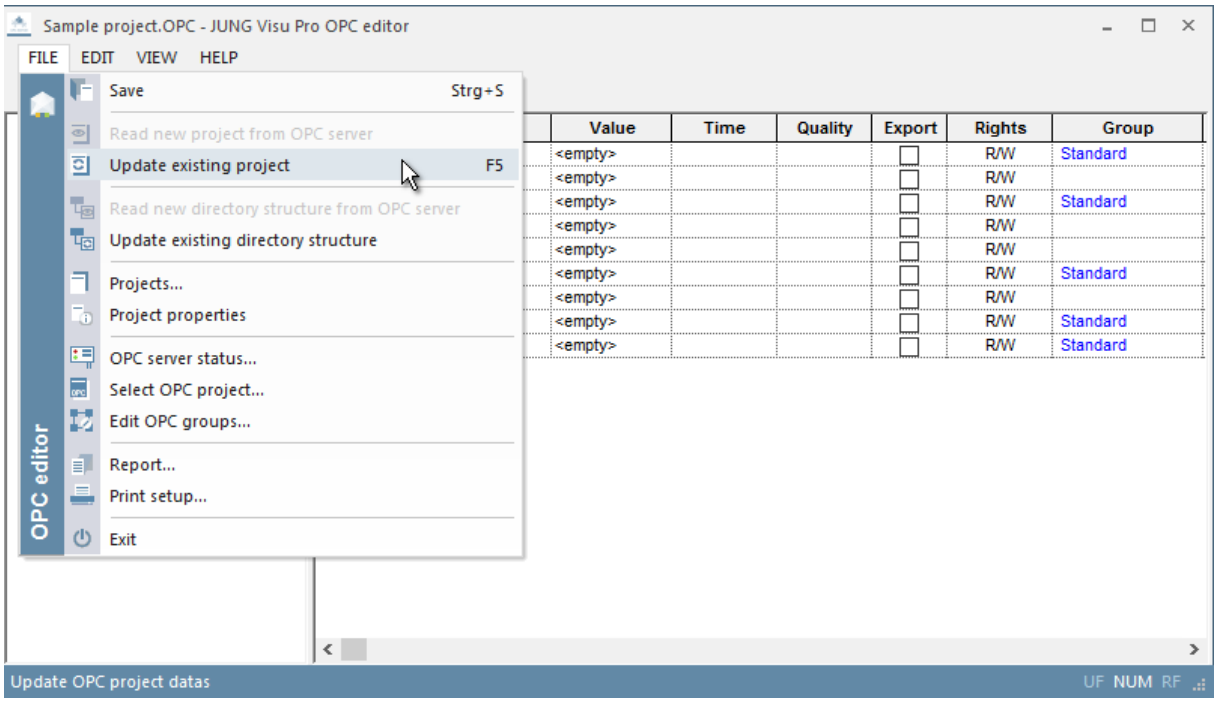


Select the desired group from the list and click the button ***OK*** to confirm.



The desired OPC group is assigned to the selected data points.

4.5 Update OPC Project



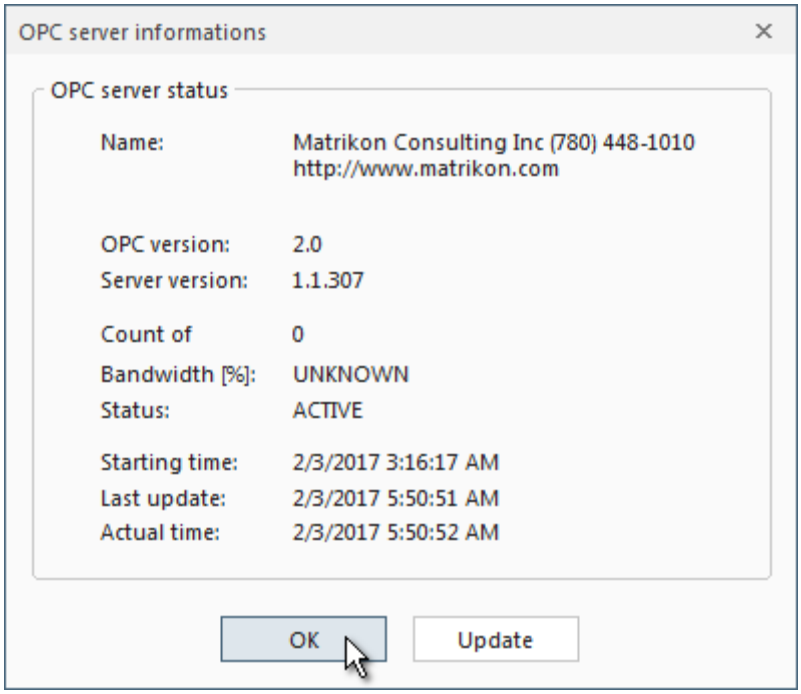
Choose the menu item ***File*** and click ***Update existing project*** to read the complete OPC project with the included data points from the server and to update them in the OPC editor (*Shortcut F5*).

To read only the existing OPC project structure, i.e. the directory tree without data points, from the server and to update it in the OPC editor, choose the menu item ***File*** and click ***Update existing directory structure***.

4.6 OPC Server

4.6.1 Server Status

Choose the menu item ***File*** and click ***OPC server status***. The dialog ***OPC server information*** opens.



The dialog displays all existing server information.

Click the button

- ***Update*** to read all OPC server information or
- ***OK*** to close the OPC server information dialog.

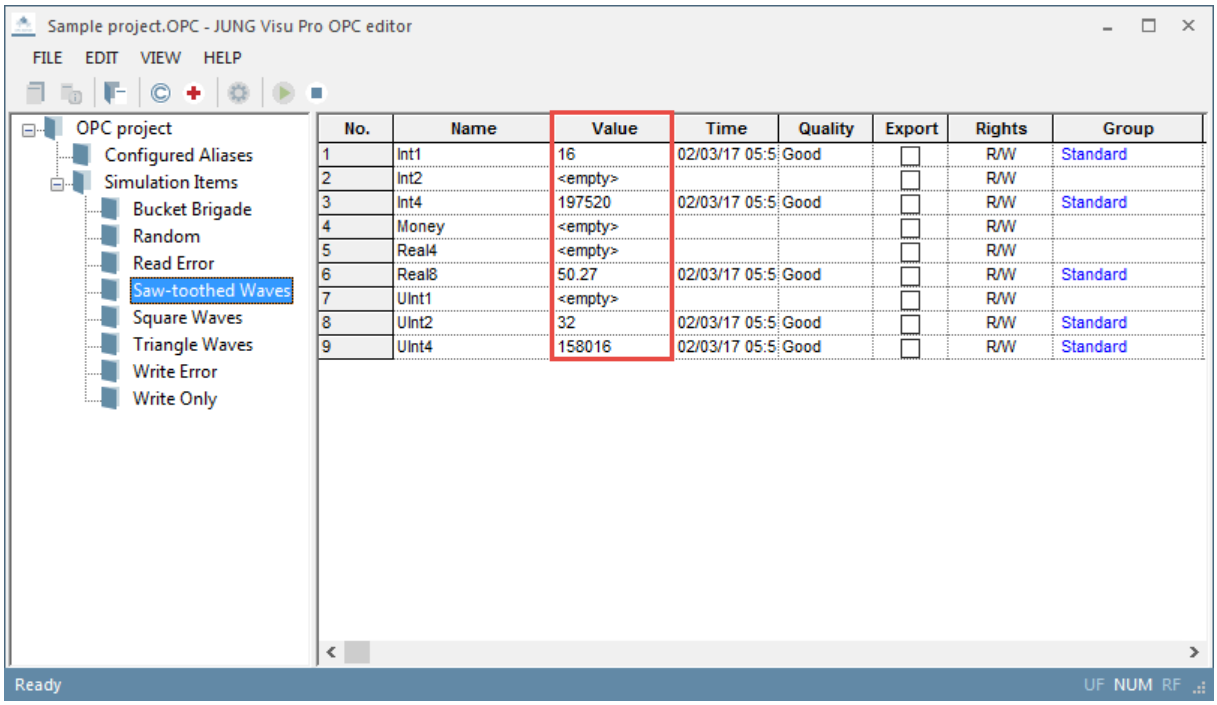
4.6.2 Start and Stop OPC Server

Start OPC Server

The process communication for updating data points is not needed and therefore it is sufficient to connect to the OPC server. To establish the connection to the selected OPC server, choose the menu item ***Edit*** and click ***Connect to OPC server***.

Or: In order to be able to send, read and write commands, the communication to the process is necessary. Choose the menu item ***Edit*** and click ***Start process communication*** or click the button in the toolbar. The connection to the selected OPC server will be produced and the process communication will be started.

The data of the OPC project will be read and updated.



Stop OPC Server

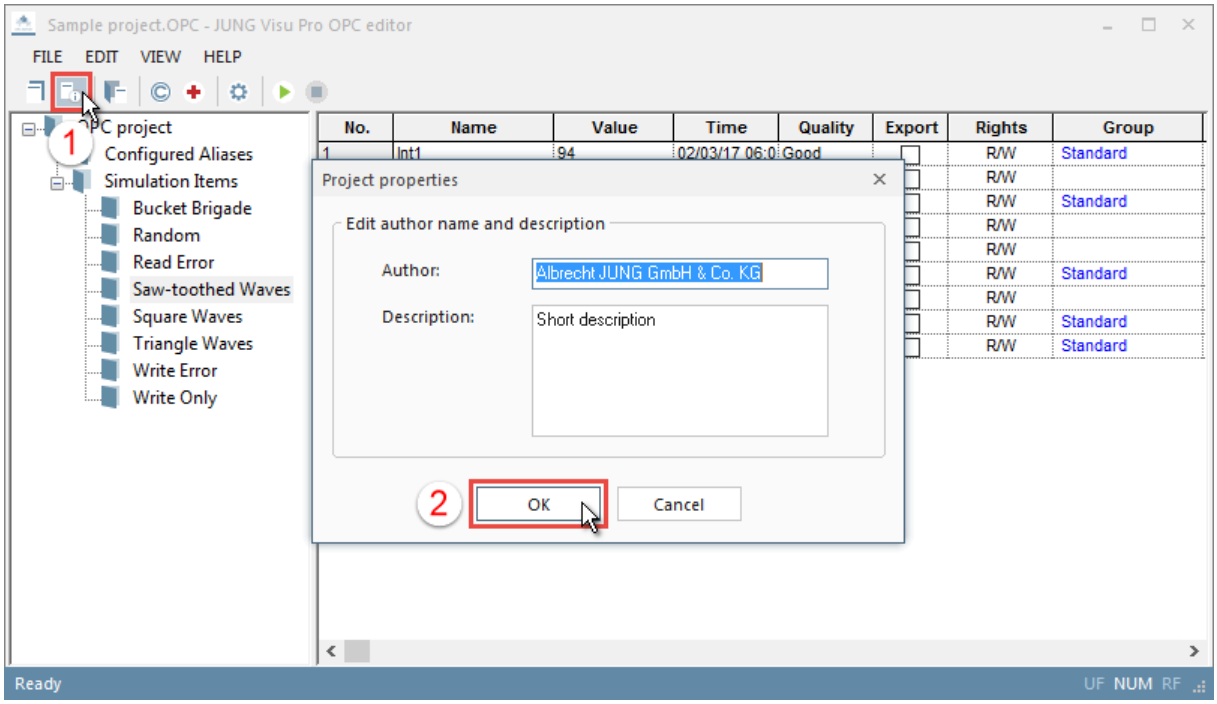
To stop the process communication, choose the menu item ***Edit*** and click ***Stop process communication*** or click the button in the toolbar.

Then choose the menu item ***Edit*** and click ***Disconnect OPC server***. The connection to the running OPC server will be interrupted.

4.7 Edit Project Properties

Choose the menu item ***File*** and click ***Project properties*** or click the button in the toolbar.

The dialog ***Project properties*** opens.

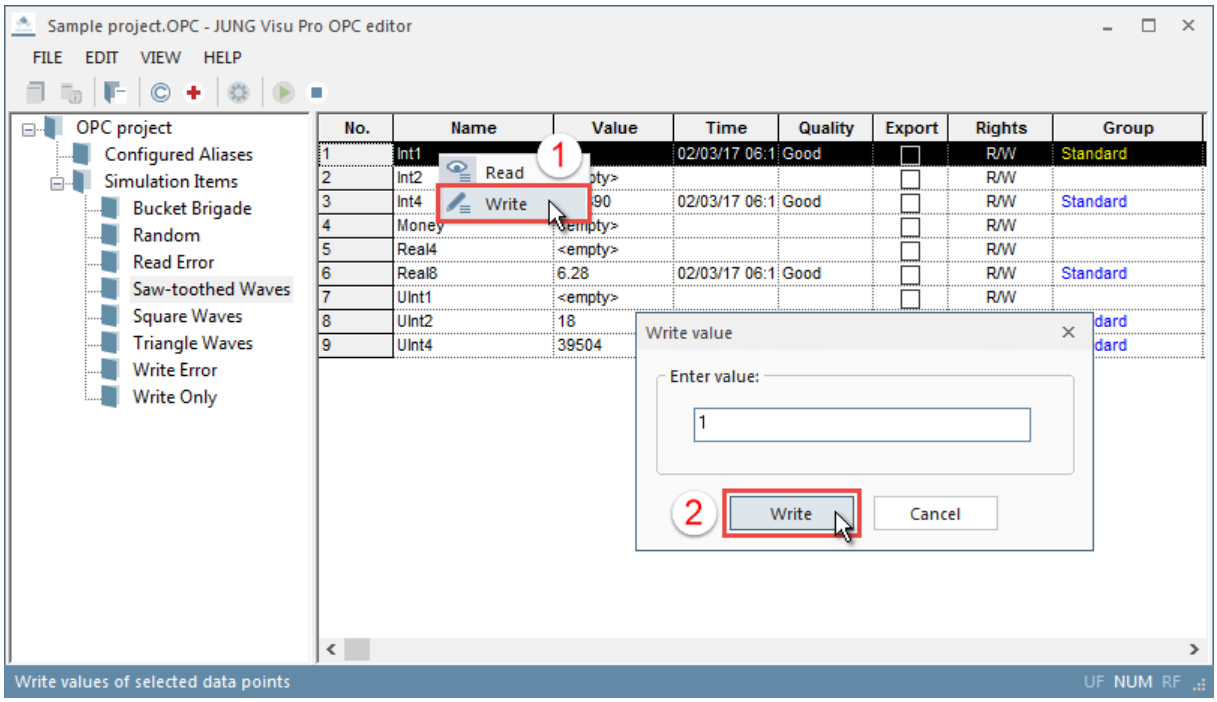


After editing the author name and the project description, click the button ***OK*** to submit the entries and to close the dialog.

4.8 Manually Send Telegrams

To send commands manually, the OPC data points must be assigned with the appropriate **access rights** and the OPC server must have been started.

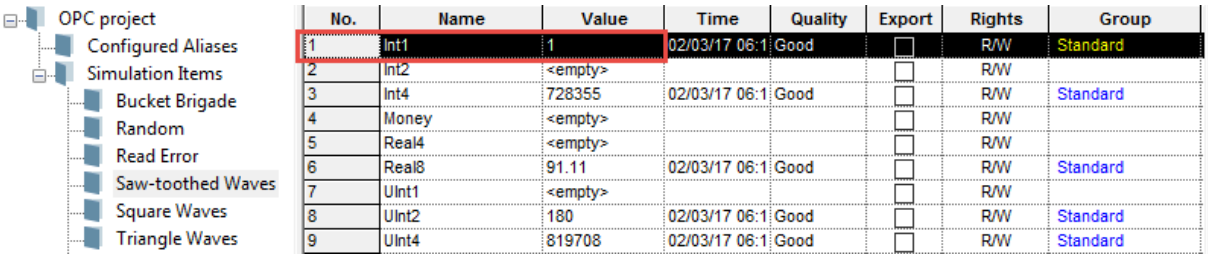
To send a value, first select the desired OPC data point (Rights: W or R/W). Then click the right mouse button on the selection to open the context menu. Alternatively, choose the menu item ***Edit*** and click ***Write value*** to open the dialog ***Write value***.



The following values can be entered:

- Binary values: for example 0 or 1 / TRUE or FALSE.
- Analog values: for example 22,00.

Click the button ***Write*** to submit the value.



When a command is sent, the contents of this column are read, converted into an OPC telegram according to the specified data point type and sent to the OPC server.

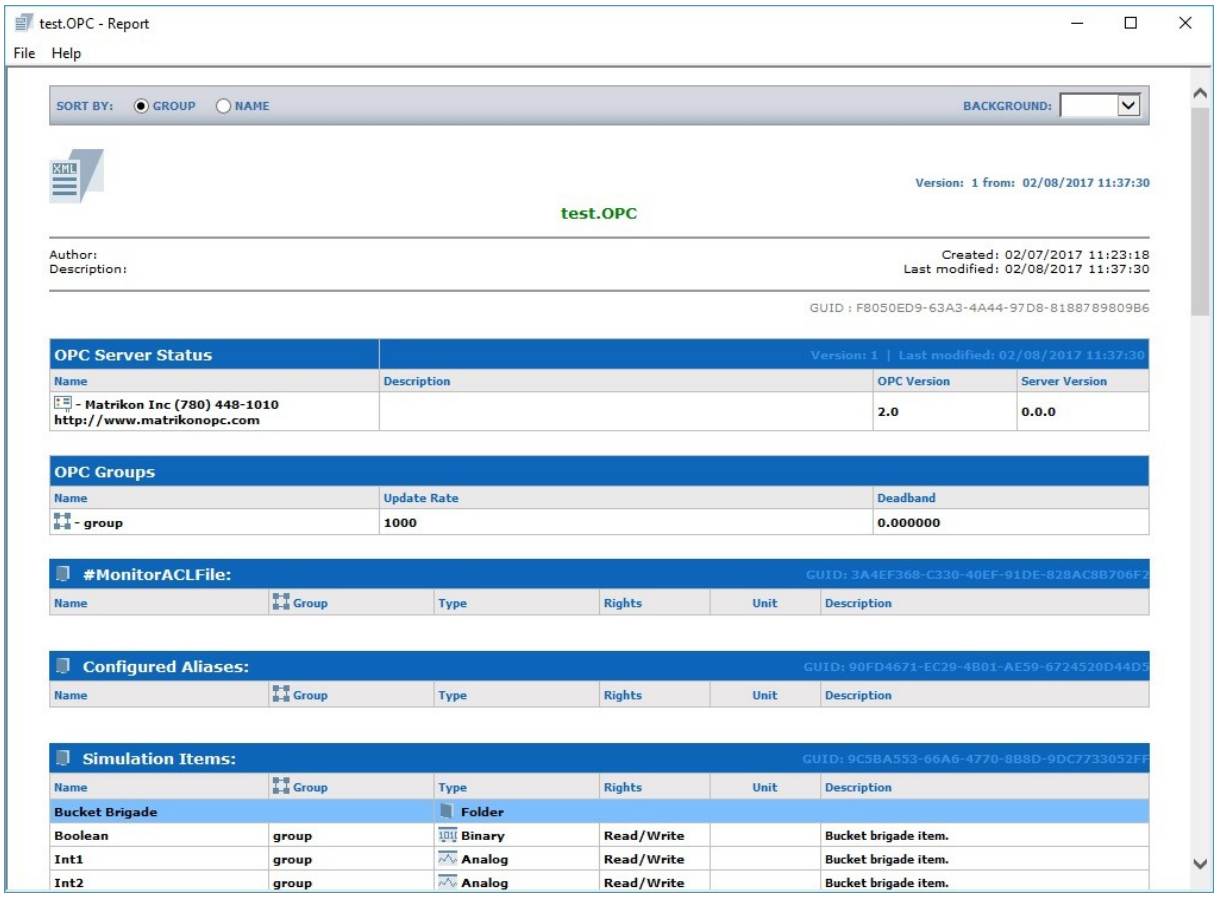
5 Reports

- Reports provide overviews of the OPC server, OPC groups and OPC data points with their properties contained in the project.
- Reports may be printed (with print preview).

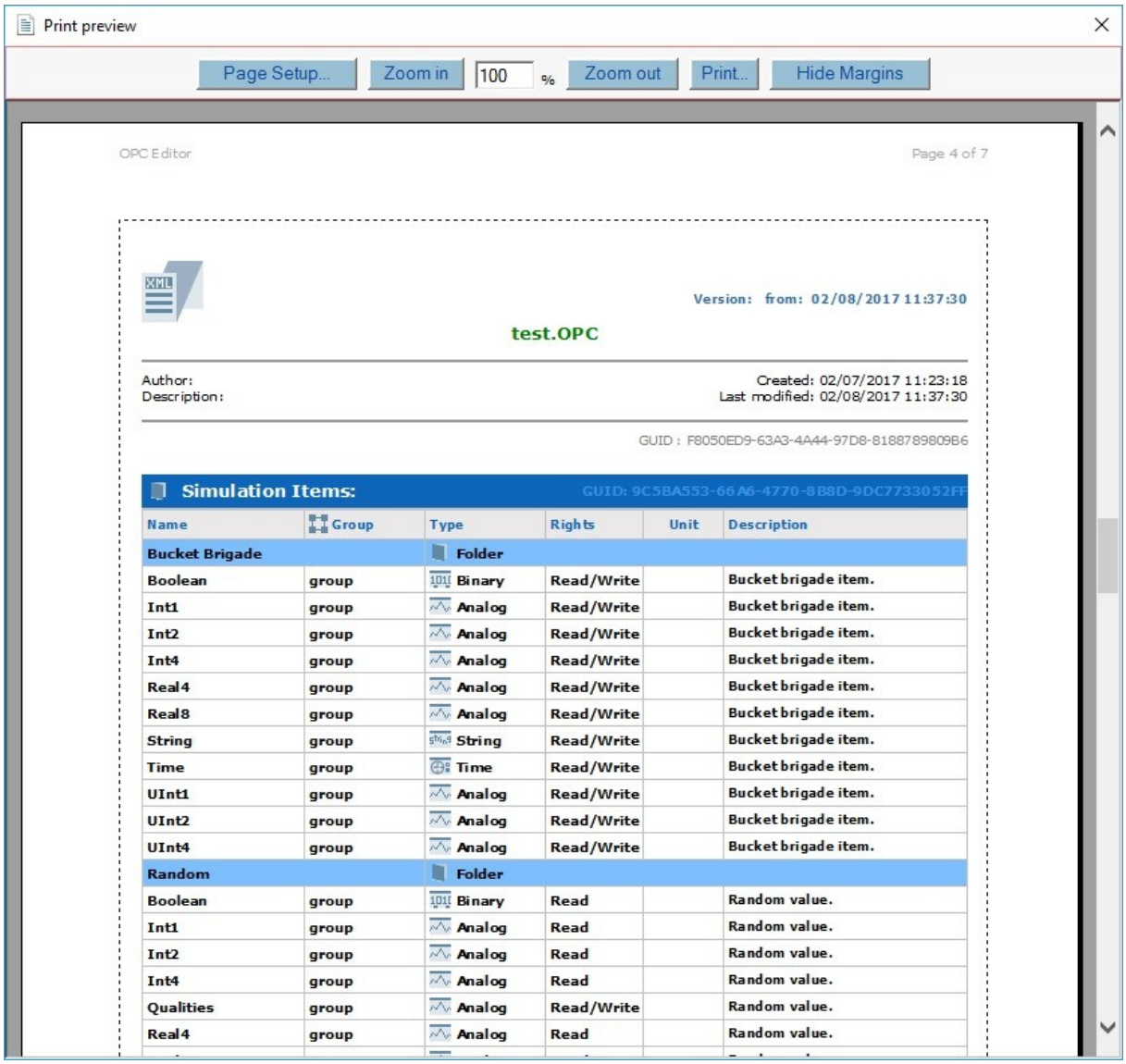
5.1 Create Report

To create a report for the current project, choose the menu item ***File*** and click ***Report***.

Use the menu at the top of the report window to sort the OPC project report by group or name.



5.2 Print Preview



Name	Group	Type	Rights	Unit	Description
Bucket Brigade		Folder			
Boolean	group	Binary	Read/Write		Bucket brigade item.
Int1	group	Analog	Read/Write		Bucket brigade item.
Int2	group	Analog	Read/Write		Bucket brigade item.
Int4	group	Analog	Read/Write		Bucket brigade item.
Real4	group	Analog	Read/Write		Bucket brigade item.
Real8	group	Analog	Read/Write		Bucket brigade item.
String	group	String	Read/Write		Bucket brigade item.
Time	group	Time	Read/Write		Bucket brigade item.
UInt1	group	Analog	Read/Write		Bucket brigade item.
UInt2	group	Analog	Read/Write		Bucket brigade item.
UInt4	group	Analog	Read/Write		Bucket brigade item.
Random		Folder			
Boolean	group	Binary	Read		Random value.
Int1	group	Analog	Read		Random value.
Int2	group	Analog	Read		Random value.
Int4	group	Analog	Read		Random value.
Qualities	group	Analog	Read/Write		Random value.
Real4	group	Analog	Read		Random value.

The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these , margin markers the user has visual control over the margin settings.

JUNG Visu Pro
instalight Editor

Content:

1 TECHNICAL REQUIREMENTS....4

1.1 Hardware....4
1.2 Operation Systems....4
1.3 Process Interface....4
1.4 Import from ETS....4
1.5 Internet Explorer, DirectX and .Net Framework....4

2 INTRODUCTION....5

3 USER INTERFACE....6

3.1 Menu....6
3.2 Toolbar....10

4 FUNCTIONS....10

4.1 instalight control Project Organizer....10
4.1.1 Create instalight control Project....11
4.1.2 Open instalight control Project....12
4.1.3 Delete instalight control Project....12
4.1.4 Rename instalight control Project....13
4.2 Edit Project Properties....14
4.3 Group Addresses....15
4.3.1 Create Main Group....16
4.3.2 Create Middle Group....16
4.3.3 Create Group Address Range....17
4.3.4 Create Group Addresses....19
4.3.5 Edit Group Addresses....20
4.4 Start Groups....22
4.5 Request Groups....25
4.6 Report Groups....29
4.6.1 Report formatting....33
4.7 instalight control KNX System Time....34
4.8 Application of KNX Data Types....35
4.9 ETS Import of Group Addresses....38
4.9.1 Create ETS Group Addresses Print Report....39
4.9.2 Create OPC Export File....39
4.9.3 Import of ETS Export Files into the instalight editor....41
4.9.4 ETS-Import Settings....44
4.10 Import Of Smart Assistant Address Files....46
4.11 Activate Address scheme....46
4.12 instalight control KNX Settings....48
4.13 Connect to instalight control KNX Bus....50
4.14 Send Telegrams Manually....50
4.15 Message List....53
4.16 Logging instalight control KNX Messages into Archives....54
4.17 Data Cache Settings....55

5 REPORTS.....56
5.1 Create Report.....56
5.2 Print Preview.....58

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz
Main memory	1024 MB
Free disk space (additional disk space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Process Interface

The connection to the instalight control KNX bus requires the installation of a suitable version of the FALCON driver of the KNX Association and the appropriate interfaces.

Optionally, the visualization system is equipped with an OPC (OLE for Process Control) client, so that instead of or additionally to KNX, OPC servers — which are available for a multitude of automation systems — can be used for the connection to the process.

1.4 Import from ETS

ETS4	YES, use of ETS, project export
ETS4	YES, use of ETS, project export
ETS3, ETS2 version 1.3	YES, use of ETS, OPC export
ETS2 versions 1.1, 1.2 and 1.3	YES, use of print reports redirected to file
Earlier ETS versions	NO

1.5 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

Click the program symbol to start the control panel of the **visualization system**. In order to clearly distinguish between the different applications, different colors were used. The icon of the **instalight editor** contains a blue colored bar . It can only be seen in the panel of the JUNG Visu Pro in the expert mode (view > expert mode).

To start the individual function modules, double-click the symbol of the respective modules.

The **instalight editor** is part of the **visualization system**. It is used to create instalight control projects which serve as a basis for the communication with the bus and for the operation of visualization projects.

First steps:

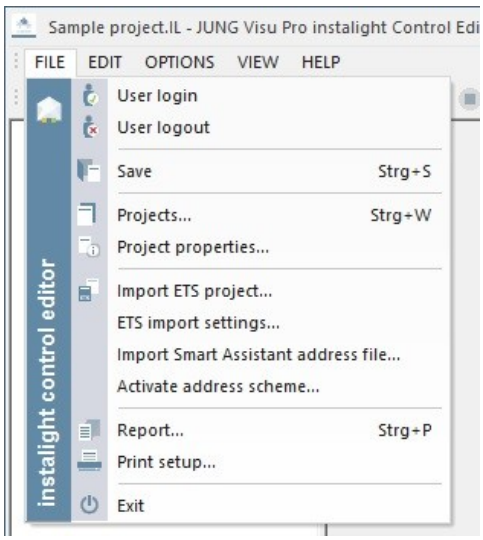
1. Create an instalight control project.
2. Create group addresses manually or import them from an ETS project.
3. Specify information concerning the data points: data point type, unit and rights.
4. Save the settings.
5. Connect the project to the instalight control KNX bus.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

3 User Interface

3.1 Menu

Menu item *File*



User login: Opens a dialog to log in a user to the system.

User logout: Logs current user out of the system.

Save: Saves the project.

Projects: Opens a dialog to create and maintain instalight control projects.

Project properties: Opens a dialog where you can edit general project properties.

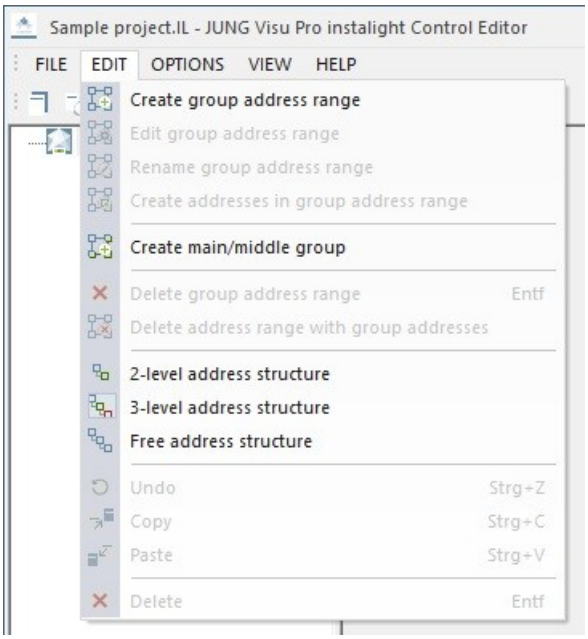
Import ETS project: Opens a dialog to import an ETS project.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Quits the instalight editor. Shortcut ALT+F4.

Menu item *Edit*



Create group address range: Opens the dialog ***New address range***.

Edit group address range: Opens the dialog for editing the range limits of the selected group.

Rename group address range: Opens the input field of the selected group address range.

Create addresses in group address range: Opens the dialog ***Create group addresses***.

Create main/middle group: Opens the dialog ***New main group*** or ***New middle group***.

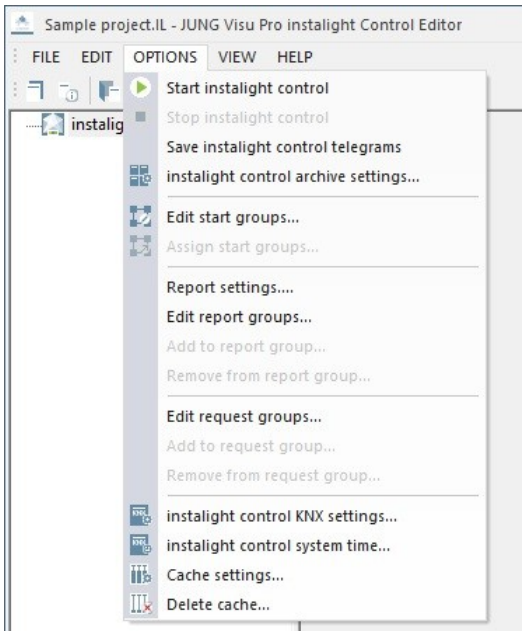
Delete group address range: Deletes only the selected address range. The group addresses contained in this address range will be placed in the address range above. If there is none, the group addresses will be deleted as well.

Delete address range with group addresses: Deletes the selected address range with all address ranges and group addresses included in that address range.

2-level address structure: Switches to the 2-level address structure for group addresses (00/0001).
3-level address structure: Switches to the 3-level address structure for group addresses (00/0/001).
Free address structure: Switches to the free address structure for group addresses (00001).

Standard editing options: Undo, Copy, Paste, Delete.

Menu item *Options*



Start/stop instalight control: Connects to or disconnects from the instalight control KNX bus.

Save instalight control telegrams: Saves instalight control KNX telegrams to a file.

i**nstalight control archive settings...:** Opens a dialog to edit archive settings.

Edit start groups...: Opens the dialog to edit start groups.

Assign start groups: Opens the dialog to assign start groups.

Report settings: Opens the dialog to edit the report groups.

Edit report groups: Opens the dialog to edit report groups.

Add to report group: Opens the dialog for the selection of report groups.

Remove from report group: Opens the dialog for the selection of report groups.

Edit request groups: Opens the dialog to edit request groups.

Add to request group: Opens the dialog for the selection of request groups.

Remove from request group: Opens the dialog for the selection of request groups.

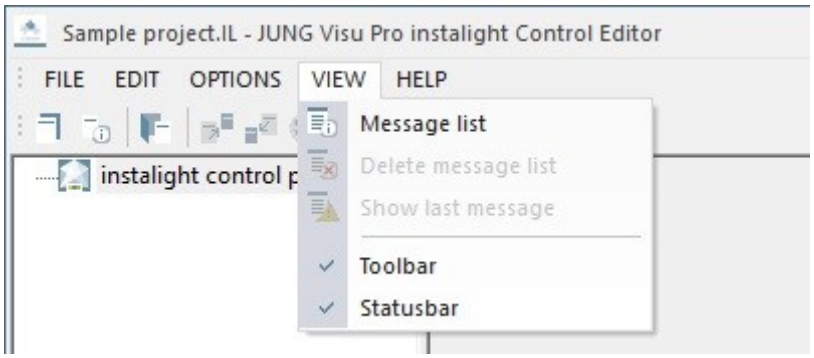
instalight control KNX settings: Opens the dialog ***instalight control KNX project settings***.

instalight control system time: Opens the dialog ***Bus time***.

Cache settings: Opens a dialog to configure the data cache.

Delete cache: Deletes the cache.

Menu item *View*

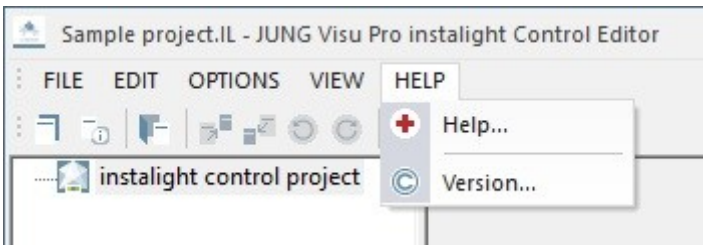


Message list: Toggles the visibility of the messages list at the bottom of the editor.

Delete message list: Deletes the messages from the message window. This operation does not affect the saving of telegrams to an archive file.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

Menu item *Help*



Content: Opens help file.

Version: Shows version and copyright information.

3.2 Toolbar

Projects management: Opens a dialog to create and maintain instalight control projects.

Project properties: Opens a dialog to edit general project properties.

Save: Saves the project.

Copy: Copies selected items to clipboard.

Paste: Pastes items from clipboard.

Undo: Undo last action.

Redo: Redo last action, i.e. reverse an undo operation.

Version: Shows version and copyright information.

Help: Opens help file.

Messages: Toggles visibility of message list at the bottom of the editor.

Start instalight control connection: Connects to instalight control KNX bus.

Stop instalight control connection: Disconnects from instalight control KNX bus.

4 Functions

4.1 instalight control Project Organizer

In the dialog ***instalight control project organizer*** you can create new projects and open, close, delete or rename existing projects.

In the project organizer you can use the buttons on the sidebar or the commands under the menu item ***Project***.

The project organizer provides the following options:

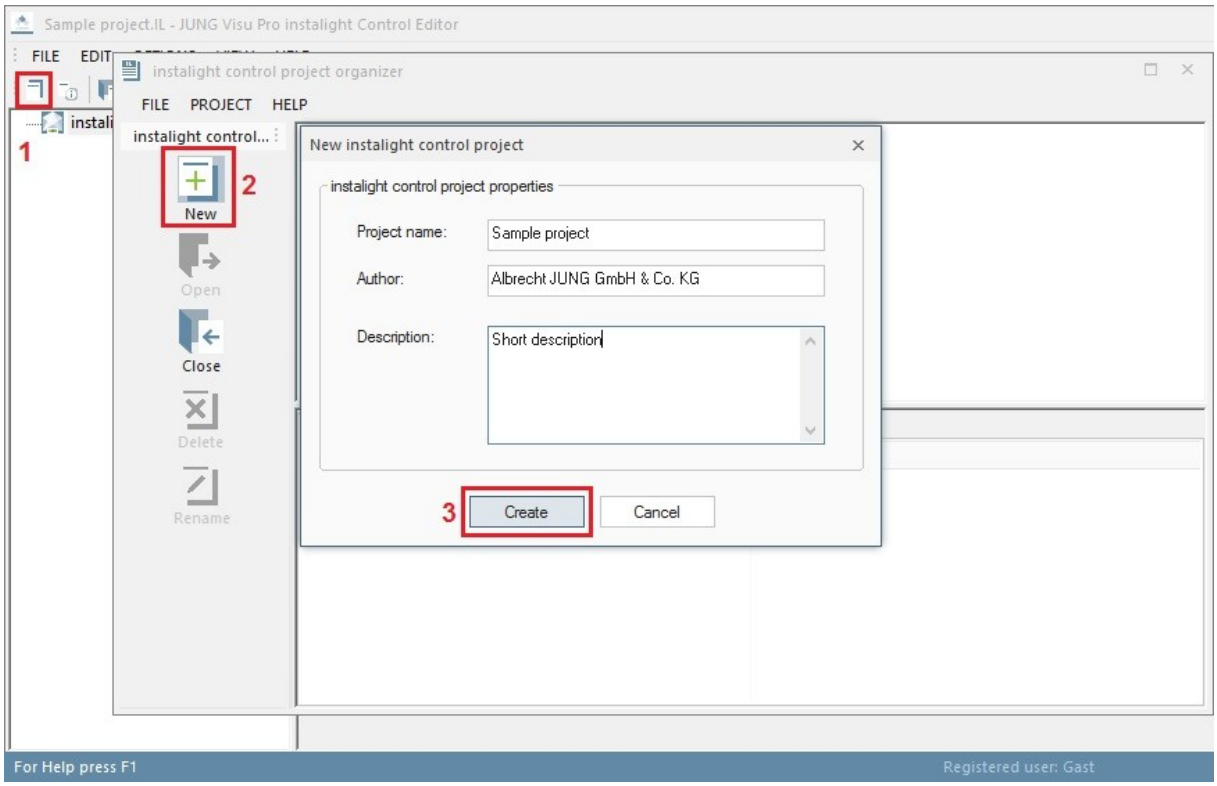
- **New**: Create a new instalight control project.
- **Open**: Open an existing project in the instalight editor.
- **Close**: Close the currently opened project.
- **Delete**: Delete the selected project.
- **Rename**: Rename a project.

4.1.1 Create instalight control Project

The dialog ***instalight control project organizer*** opens automatically when the instalight editor is started.

To start it manually in the instalight editor, choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

To create a new instalight control project, click the button ***New*** on the sidebar or choose the menu item ***Project*** and click ***New***. The dialog ***New instalight control project*** opens.



Enter the data of the new project:

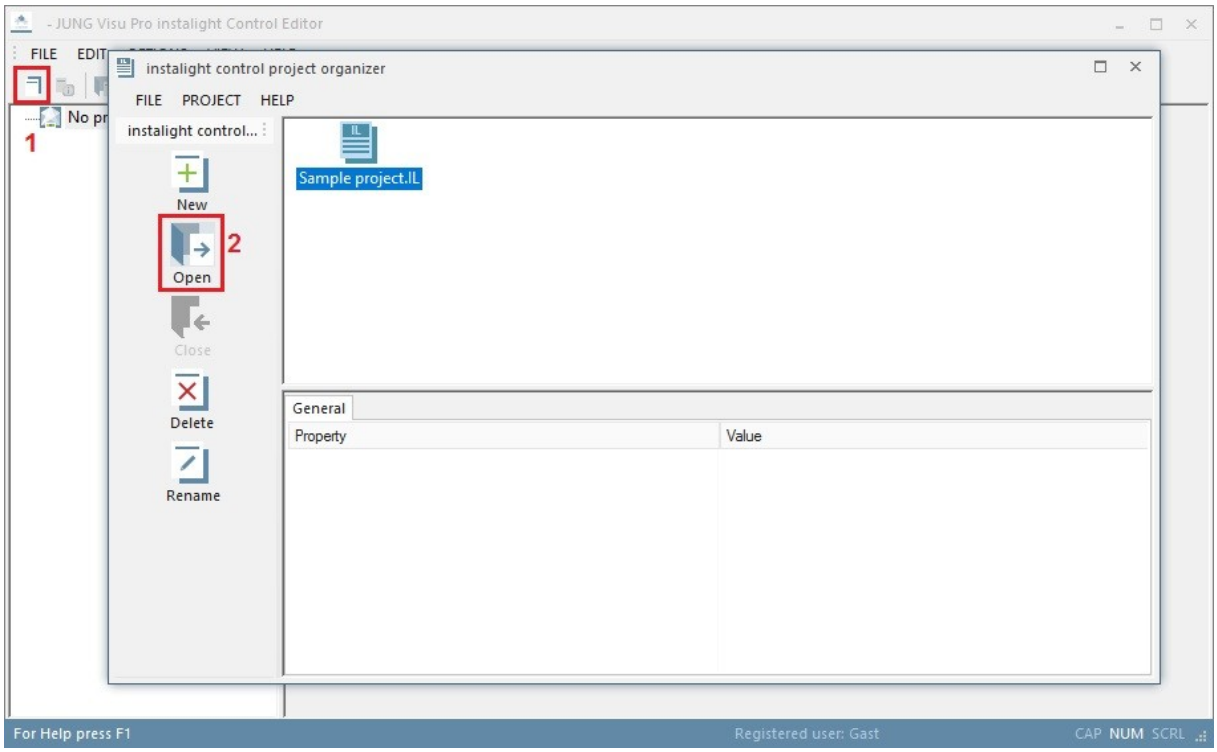
Project name: Enter a unique name for the instalight control project.

Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or edited later.

Create: Press the button ***Create*** to commit the entries and to finish the creation of the project.

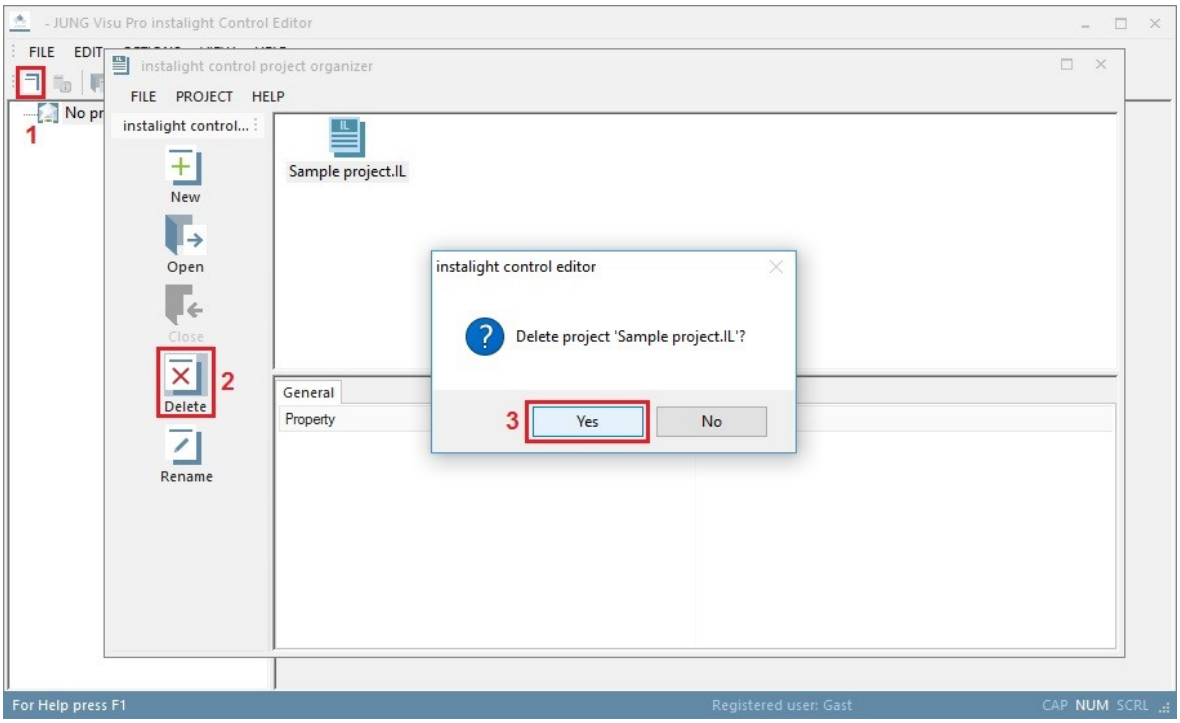
4.1.2 Open instalight control Project

To load a project into the instalight editor, double-click the project symbol in the project list or select the desired project and click the button ***Open*** in the sidebar.



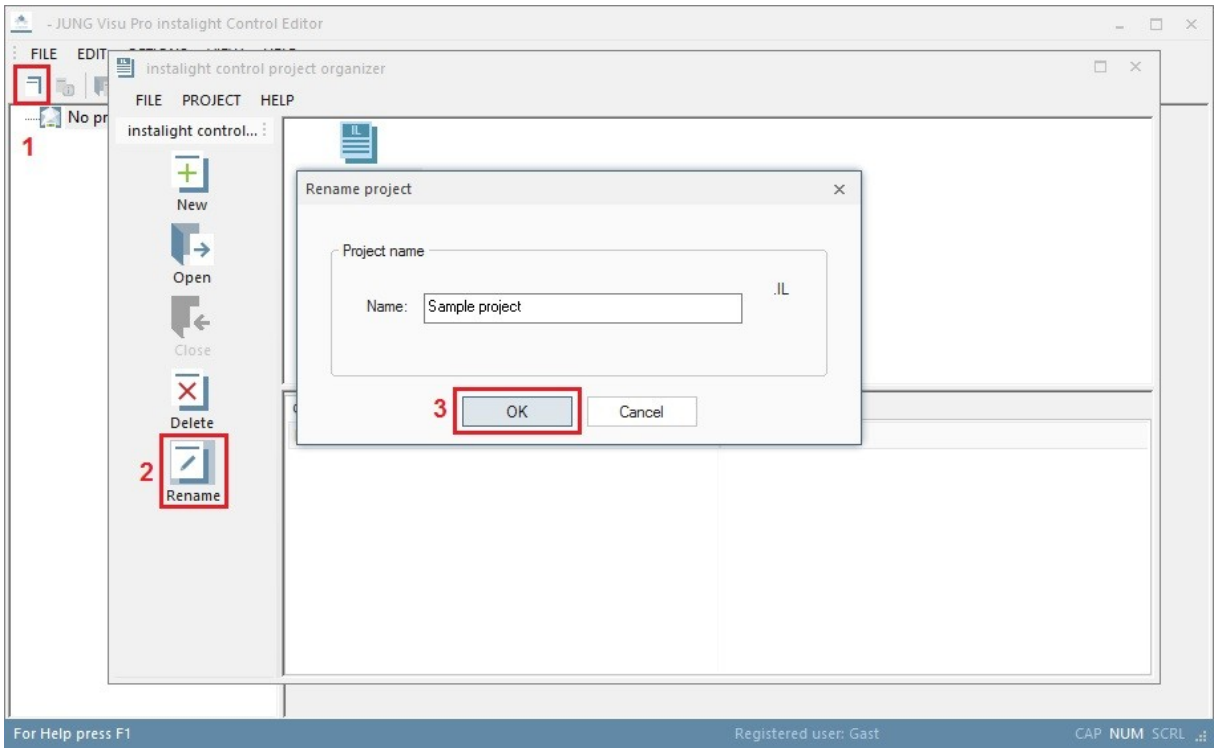
4.1.3 Delete instalight control Project

To delete a project, select the project and click the button ***Delete*** in the sidebar.



4.1.4 Rename instalight control Project

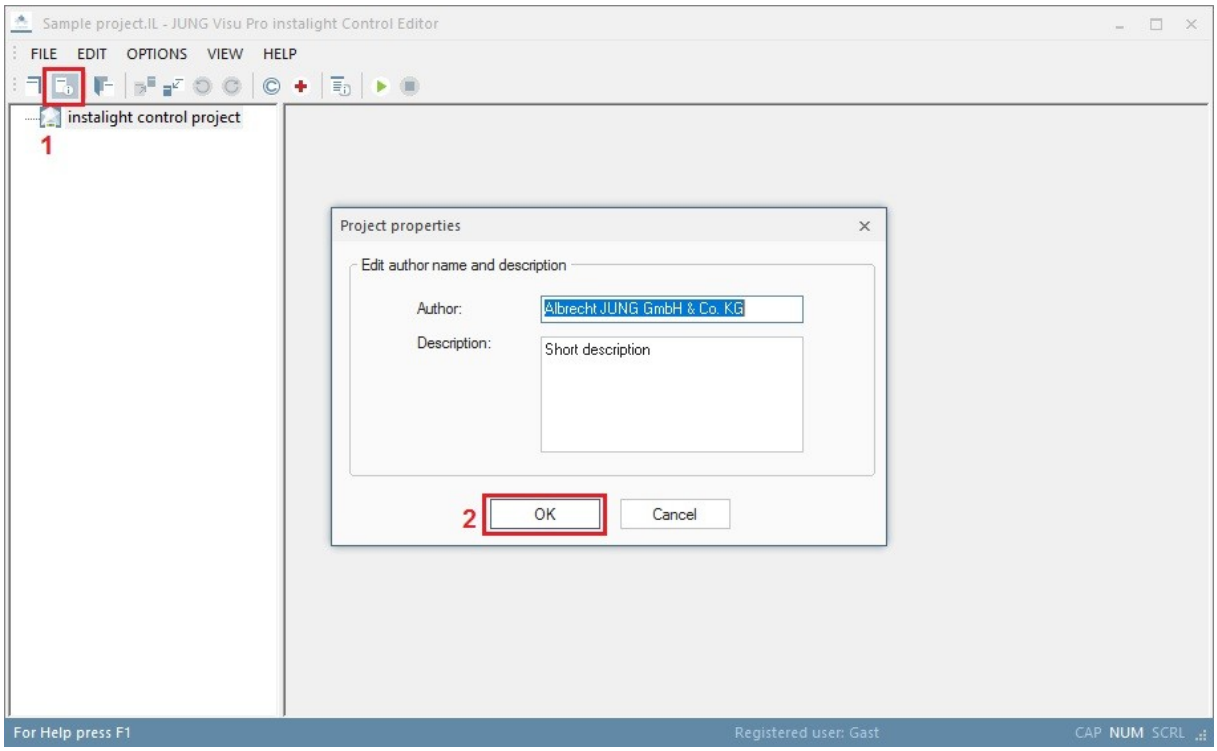
To rename a project, select the desired project and click the button ***Rename*** in the sidebar. The dialog ***Rename project*** opens.



Enter in the name field the new project name and click the button ***OK***. The dialog is closed and the new project name is adopted. The extension ***.IL*** is automatically added to the project name.

4.2 Edit Project Properties

Choose in the instalight editor the menu item ***File*** and click ***Project properties*** or click the symbol in the toolbar. The dialog ***Project properties*** opens where you can edit general properties of the project.

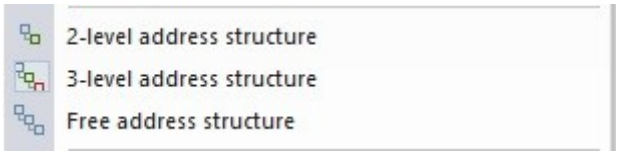


Edit the specifications author and description and click the button ***OK*** to commit the changes and to close the dialog.

4.3 Group Addresses

You can create group addresses manually or import them from ETS projects via the import function. More details on the ETS import will be provided in one of the subsequent chapters. In the following, the manual creation of group addresses is described.

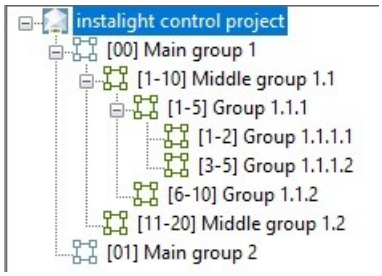
Under the menu item ***Edit*** you can choose between 3 different structures of group addresses:



2-level address structure: The group addresses are arranged in a 2-level scheme; main group address and group address *(00/0001)*.

3-level address structure: The group addresses are arranged in a 3-level scheme; main group address, middle group address and group address *(00/0/001)*.

Free address structure: Free display of the group addresses *(00001)*.

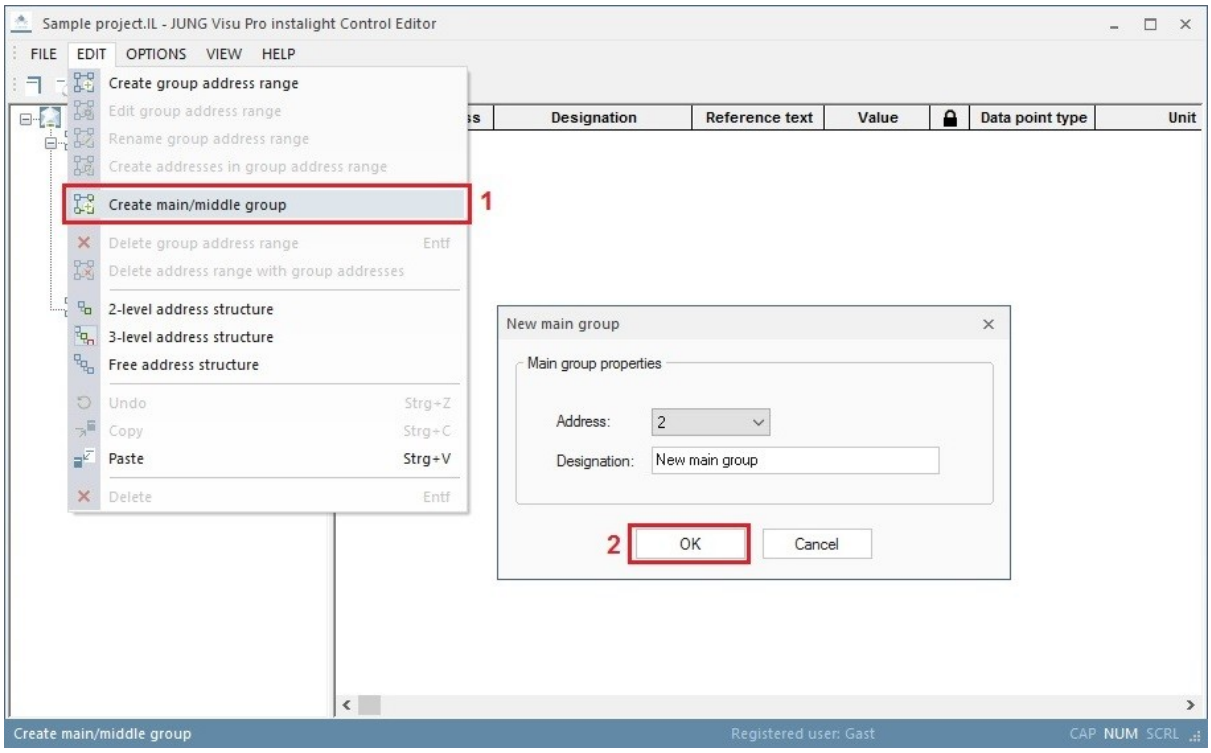


In the left window of the editor you can see the instalight control projects with their respective groups (if already created) arranged in a structure tree.

4.3.1 Create Main Group

To create a main group,

- choose the menu item ***Edit*** and click ***Create main/middle group*** or
- right-click the project symbol in the directory tree and click in the context menu the option ***Create main/middle group***.



The dialog ***New main group*** opens. Enter the main group address and a name for the main group. Click the button ***OK*** to commit the data and to close the dialog.

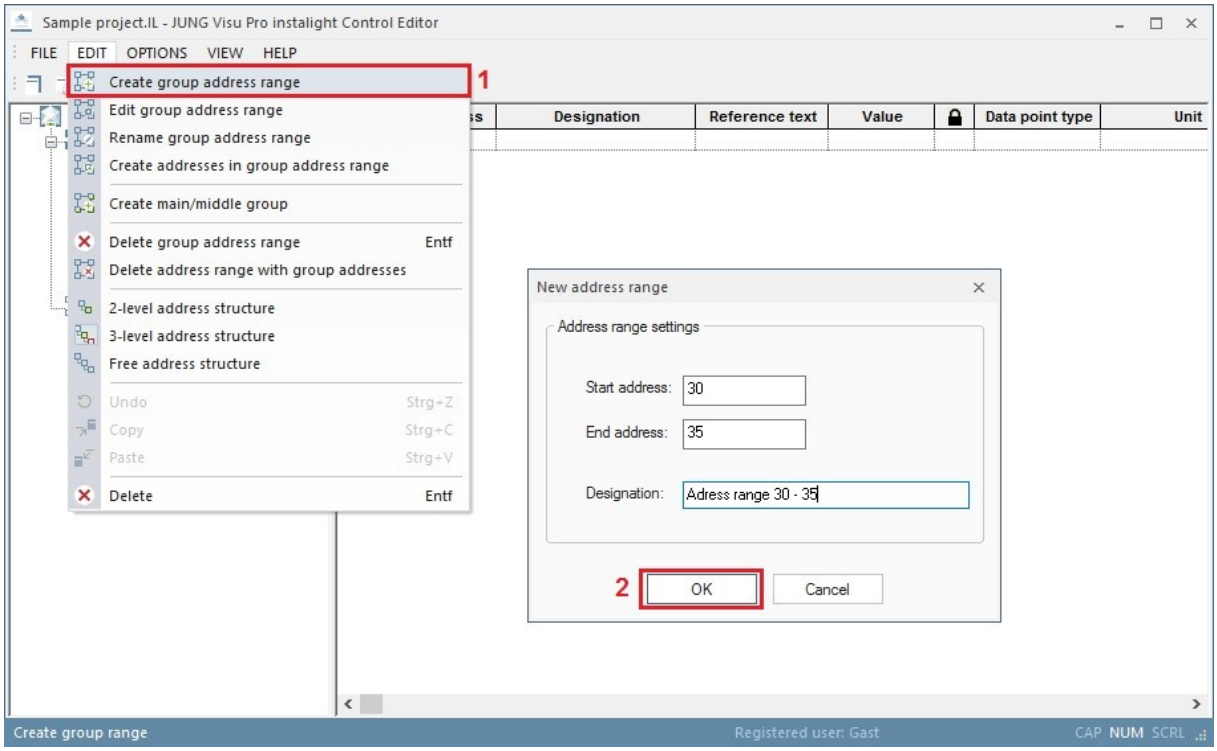
Use this method to create further main group addresses.

4.3.2 Create Middle Group

To create a middle group, choose the menu item ***Edit*** and click ***Create main/middle group***. Alternatively, right-click the main group symbol in the directory tree and click in the context menu the option ***Create main/middle group***.

4.3.3 Create Group Address Range

To create a group address range, select in the directory tree the project or group for which you want to create the address range. Choose the menu item ***Edit*** and click ***Create group address range***. Alternatively, right-click the project or group in the directory tree and click in the context menu the option ***Create group address range***.



The dialog ***New address range*** opens. Enter the start and the end address (range limits) and, where appropriate, a name for the selected group address range. Click the button ***OK*** to commit the entry.

The specified address range will then be assigned to the corresponding or the superordinate address range.

Change group address range:

Select in the directory tree the group address range you wish to edit. Choose the menu item ***Edit*** and click ***Edit group address range***. A dialog opens where you can change the range limits of the selected group address range. Enter the new range limits and click the button ***OK*** to commit the change.

Rename group address range:

Select in the directory tree the group address range you wish to rename. Choose the menu item ***Edit*** and click ***Rename group address range***. The input field of the selected group address range opens. Enter the new name and press the **enter** key.

Delete group address range:

Select in the directory tree the group address range you wish to delete. Choose the menu item ***Edit*** and click ***Delete group address range***. Answer the confirmation prompt with ***Yes*** to delete the selected address range. The contained ranges and group addresses will be placed in the superordinate group address range. If there is no superordinate range, the group addresses will be deleted as well.

Delete group address range with group addresses:

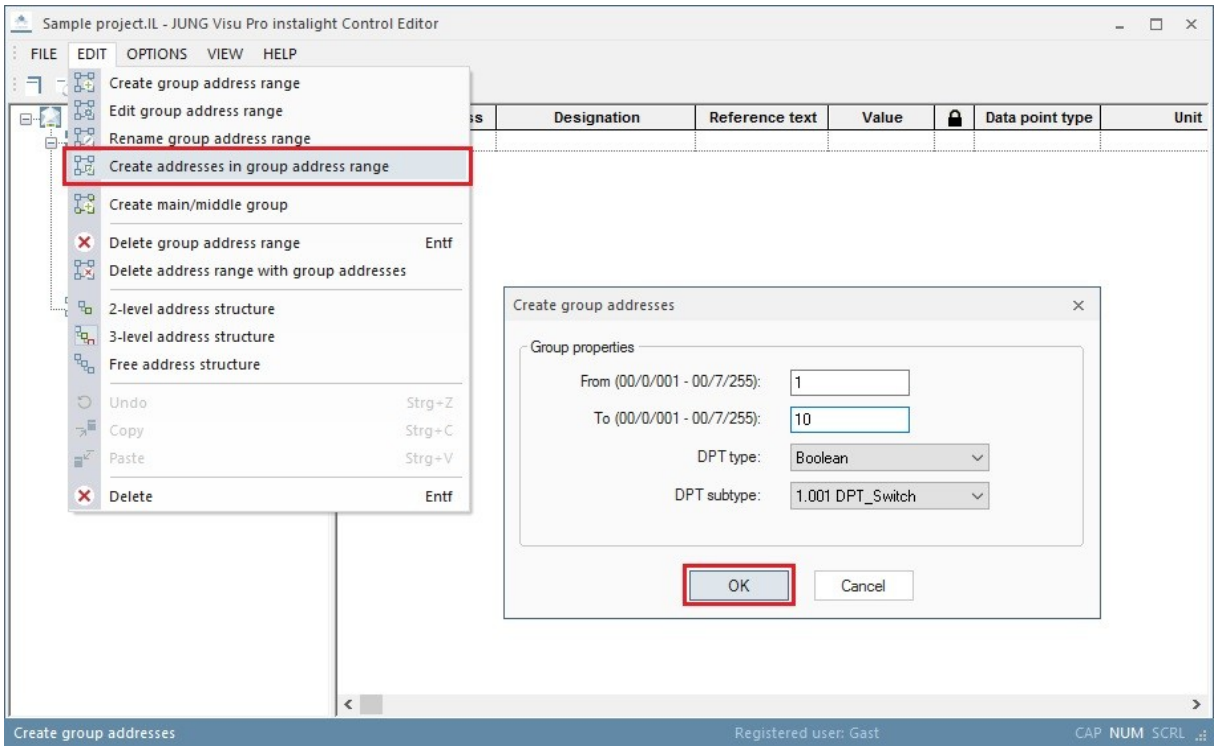
Select the group address range in the directory tree. Choose the menu item ***Edit*** and click ***Delete address range with group addresses***. Answer the confirmation prompt with ***Yes*** to delete the selected address range along with all ranges and group addresses contained in it.

4.3.4 Create Group Addresses

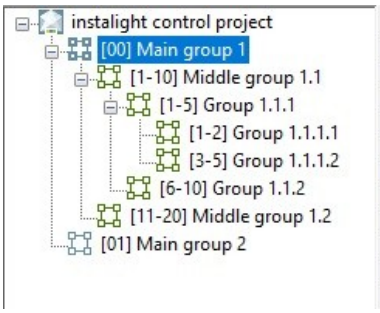
To create group addresses in a group address range,

- select the group address range, choose the menu item ***Edit*** and click ***Create addresses in group address range*** or
- right-click the group address range and click in the context menu the option ***Create addresses in group address range***.

The dialog ***Create group addresses*** opens.



The entry fields ***From*** and ***To*** allow to enter several group addresses in the selected address range, having the same data point type. If you already know the number of the required group addresses, you can create them all at once and configure them manually in the fields afterward. Here you can also change the middle group address.



No.	Address	Designation
1	00/0/001	Group 00/0/001
2	00/0/002	Group 00/0/002
3	00/0/003	Group 00/0/003
4	00/0/004	Group 00/0/004
5	00/0/005	Group 00/0/005
6	00/0/006	Group 00/0/006
7	00/0/007	Group 00/0/007
8	00/0/008	Group 00/0/008
9	00/0/009	Group 00/0/009
10	00/0/010	Group 00/0/010
11		

4.3.5 Edit Group Addresses

Each group address requires further specifications to enable the instalight editor to properly interpret telegrams:

Address: You can enter an address directly in the address column. Alternatively, select the group address range in the directory tree, right-click it and click in the context menu the option ***Create addresses in group address range***.

Designation:

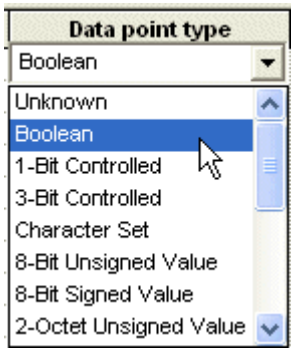
No.	Address	Designation
1	00/0001	Light 1

Enter a name for the group address. The designation serves the identification of the address during further processing of the data. The instalight editor and other parts of the system will refer to the group address by this name.

Value: If a group address receives values, they are displayed in this field. If write access is allowed, you may enter a value in this field and send a value to the group address.

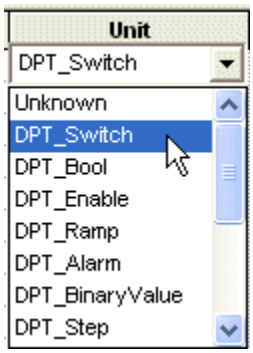
Reference text: An alternative text for the installation name which will be exported to the csv file as well.

Lock address: When this option is enabled, the data point will not be updated during an ETS import.



Data point type:

Type of the address. To assign a data point type (DPT), click in the input field and select the desired data point type.



Unit:

After a type is set, an applicable KNX unit can be chosen. The specification of the unit is **optional**. If the unit has been defined for a group address, it will be shown in the messages list in a plain text message.

To assign a unit, click in the input field and select a unit from the dropdown list. The list automatically contains only the units which can be assigned to the data point type.

Rights:

To execute the desired action via the bus, the read and write rights must be set.

None: This is the default: Neither read nor write access is allowed. It is not possible to read the group address nor to write values to it. However, spontaneous messages from this group address can be received and interpreted.

Read: It is only possible to send read commands to the group address (no write commands).

Write: It is only possible to send a value to the KNX group address. The instalight editor denies read access to the address (important for e.g. switch objects).

Read/Write: It is possible to send a value to the group address and to read it.

Initialization: Here you can assign the group address to a start group.

Description: Enter a short description of the address (optional).

Ref.unit (0): The notation of a certain state can be changed here (for example if an "off" instead of 0 is expected). This will be exported with the report data.

Ref.unit (1): The notation of a certain state can be changed here (for example if an "on" instead of 1 is expected). This will be exported with the report data.

Note: *When you move the cursor to another row, either with the mouse or keyboard keys, the input of the current KNX group address will be committed. Leaving the edited row will commit the row's data!*

A new empty row will be displayed to enter the next KNX group address.

The cursor can be moved either with mouse click on a cell or keyboard arrow keys. The field under the cursor can be edited. To move the cursor between the cells of one row, use the <tab> and <shift>+<tab> keyboard keys.

Save entries:

Before the new group address definitions can be used, the data must be saved. The editor can only interpret telegrams from the instalight control KNX bus after the newly entered data have been saved.

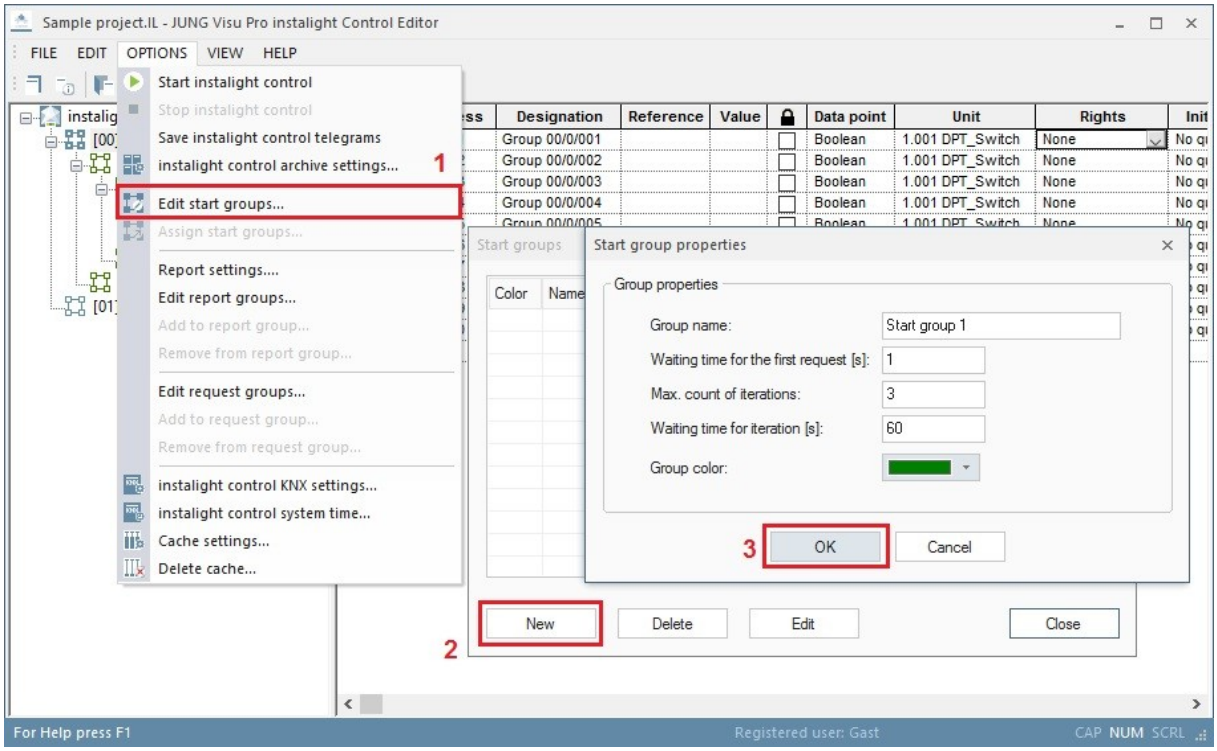
To save the project, choose the menu item ***File*** and click ***Save*** or click the symbol in the toolbar.

Warning: *Only after saving will the changes be interpreted by the bus!*

4.4 Start Groups

Choose the menu item ***Options*** and click ***Edit start groups***. The dialog ***Start groups*** opens.

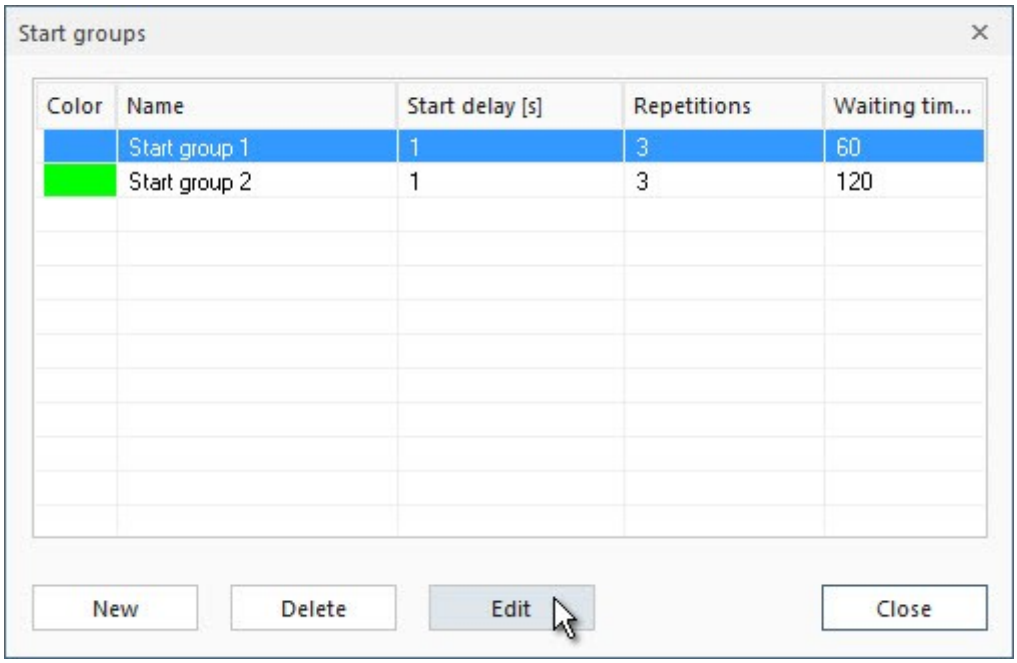
Create start group:



Click the button ***New***. The dialog ***Start group properties*** opens. Enter the settings and press the button ***OK*** to create the start group and to close the dialog. The newly created start group is now available in the dialog ***Start groups***.

Edit start group:

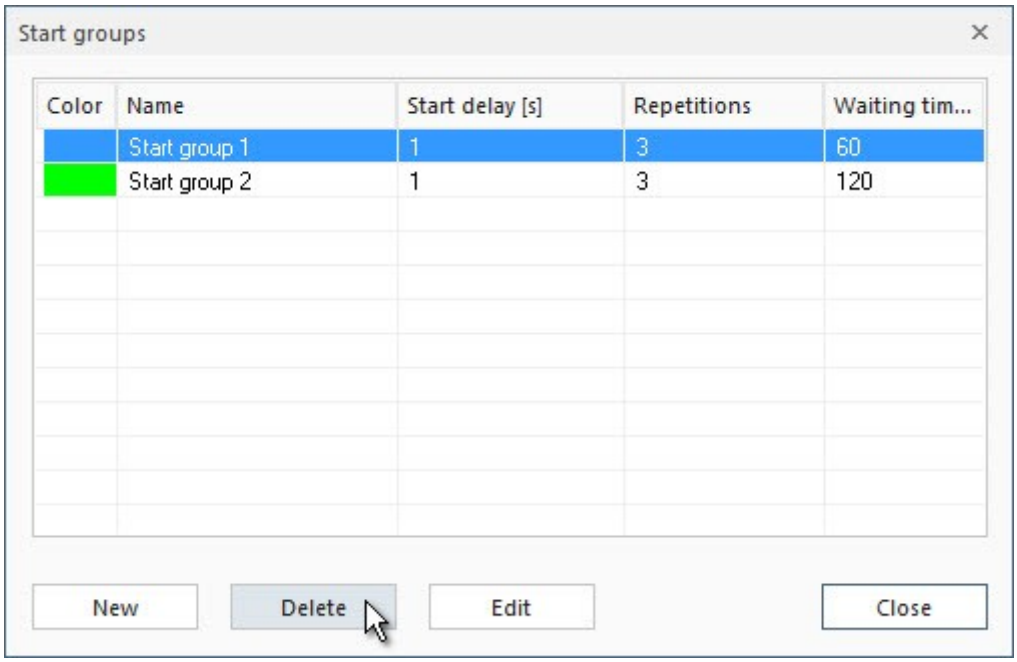
To edit the start group properties, select the start group and click the button ***Edit***. The dialog ***Start group properties*** opens.



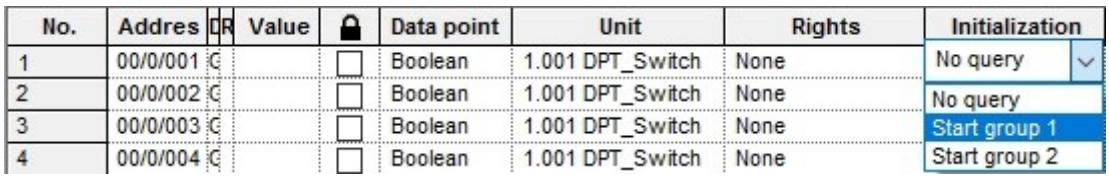
Edit the start group properties and click the button ***OK*** to commit the entries.

Delete start group:

To delete a start group, select the start group and click the button ***Delete***. Then click the button ***Close*** to quit the dialog ***Start groups***.

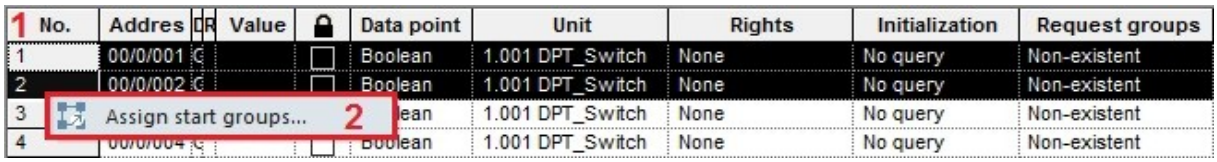


Assign start group:

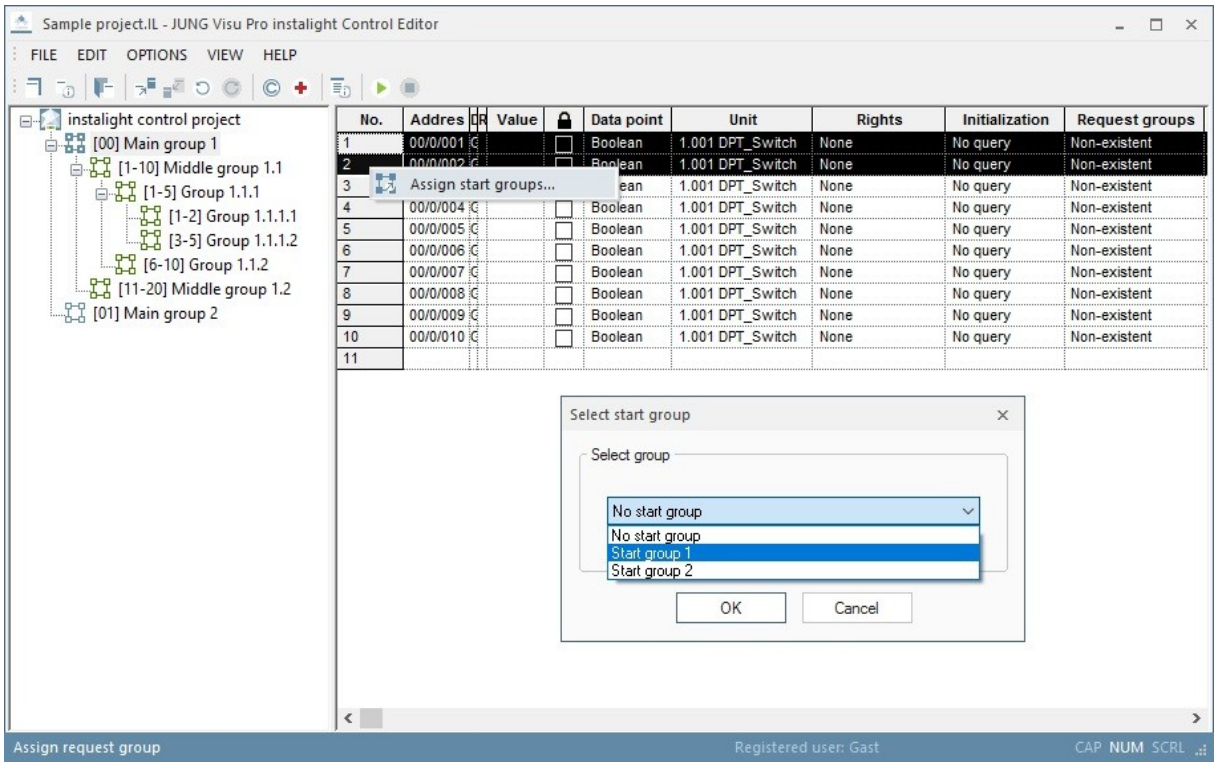


To assign a start group to a group address, click in the column ***Initialization***. Then select the start group in the combo box.

You can assign multiple group addresses to the same start group: Select the desired group addresses while pressing the control key in the first column of the schedule. Right-click to open the context menu and choose ***Assign start groups...*** The dialog ***Select start group*** opens.



Or: To assign all group addresses to a start group, click on the top left cell of the grid to select the whole table. Then right-click to open the context menu and click ***Assign start groups***. The dialog ***Select start group*** opens.



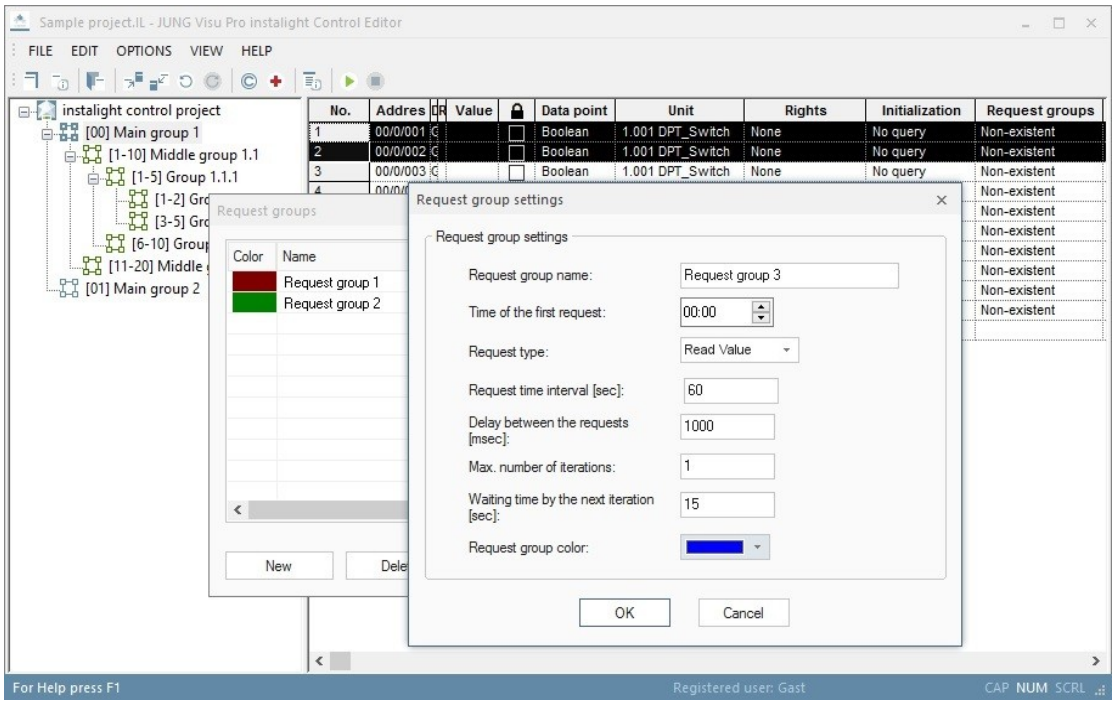
Select the desired start group and click ***OK*** to commit the entry.
All group addresses have now been assigned the start group ***Standard***.

4.5 Request Groups

Through the **Request groups**, values can be read frequently. You can enter periodically when a value should be read or you can send 1 or 0 (if the end device needs a request value before sending back its value).

Click under ***Options*** at ***Edit request groups*** to open the dialog ***Request groups***.

Create request group:



Click at ***New*** to open the dialog ***Request group settings***. Insert the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Request groups.***

Request group name: Name of the group with which the group is listed.

Time of the first request: Time between 00:00 and 23:59.

Request type: Read value / Send 1 / Send 0.

Request time interval [sec]: Time specification between 10 and 86400 seconds.

Delay between the requests [msec]: Time specification must be a value between 250 and 60000 milliseconds. This option determines the break between different request groups (necessary at a project with many data points to not overcharge the bus and lose protocols).

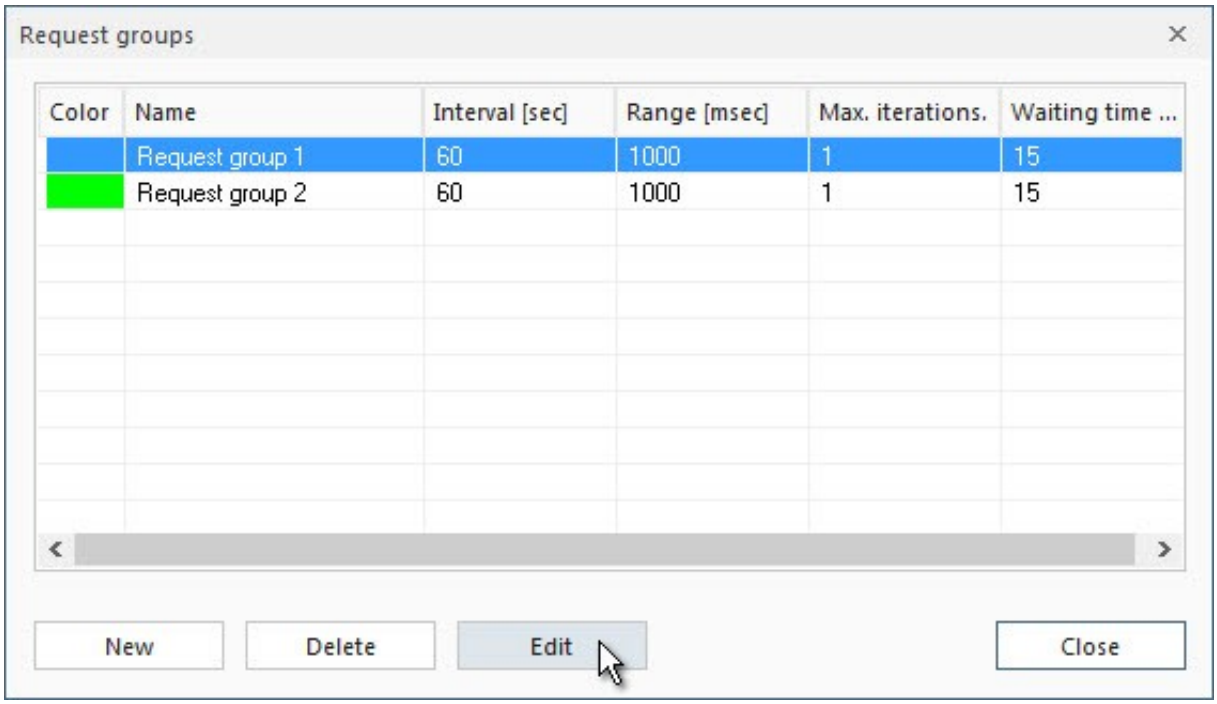
Max. number of iterations: Determines the number of repeats if the request failed.

Wating time by the next interation [sec]: Determines the time between the repeats.

Request group color: Set a color to have a better discrimination of the groups.

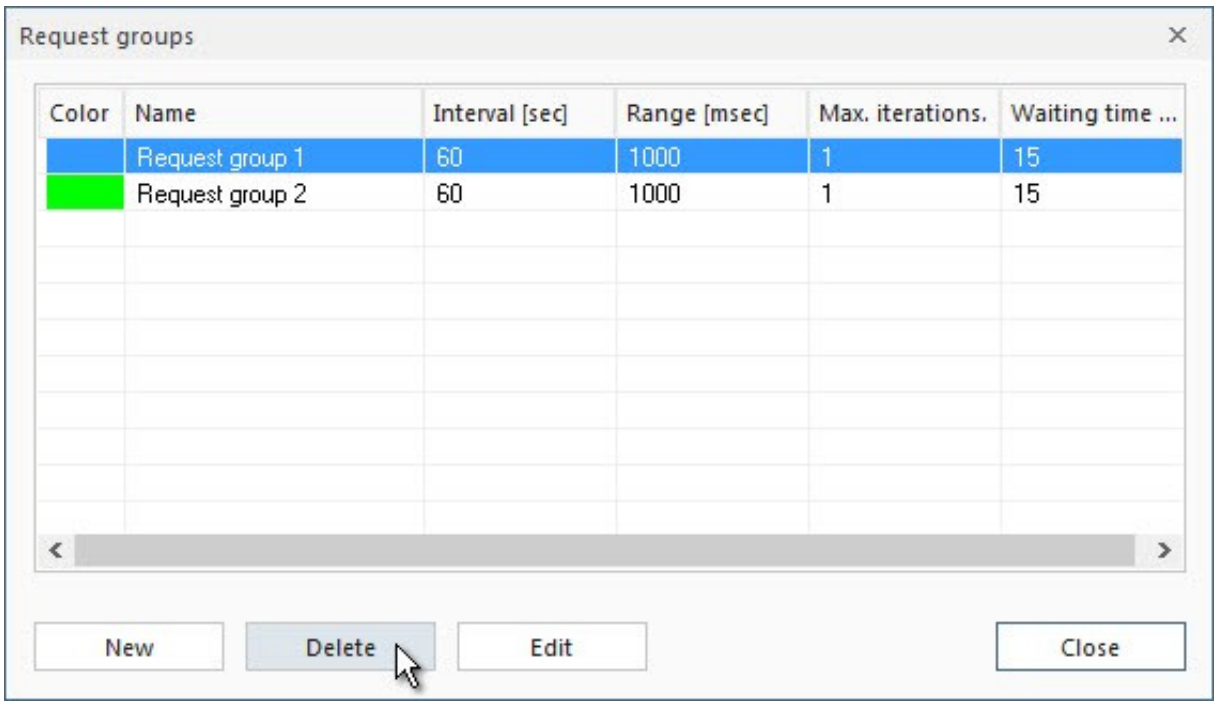
Edit request group:

To edit a request group, select the request group and click the button ***Edit***. The dialog ***Request group settings*** opens.



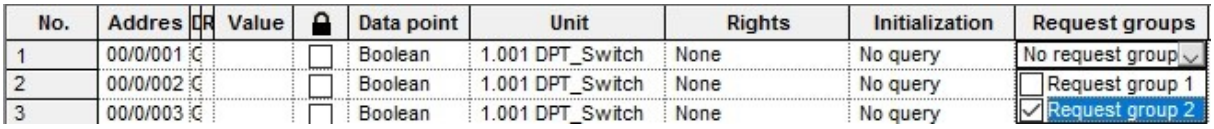
Edit the group settings and save the changes by clicking ***OK***.

Delete request group:



To delete a request group, select the request group and click the button ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Request groups***.

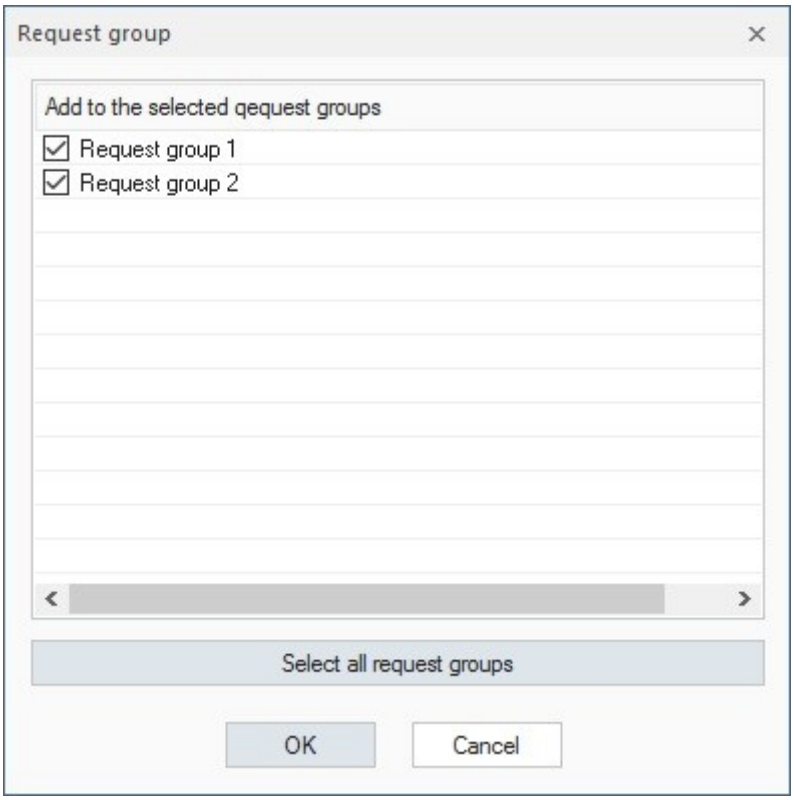
Assign request group:



To assign a request group to a group address, click in the column ***Request groups*** to select one or multiple request groups.

Or: To assign a selection of group addresses to a request group, select the first group address in the column ***Request groups*** and draw the mouse over the beneath lying lines. Right-click on the selected elements to open the context menu and choose ***Add to request group***.

The dialog ***Request group*** opens.



Set a checkmark at the desired request groups and confirm with ***OK***.

No.	Value		Data point	Unit	Rights	Request groups
1		☐	Boolean	1.001 DPT_Switch	None	2 request groups
2		☐	Boolean	1.001 DPT_Switch	None	2 request groups
3		☐	Boolean	1.001 DPT_Switch	None	Request group 1
4		☐	Boolean	1.001 DPT_Switch	None	Request group 2

For every group address is the request group now visible or, if a group address has multiple request groups, the number of request groups.

Delete request group:

Deleting request groups is analog to assigning request groups.

4.6 Report Groups

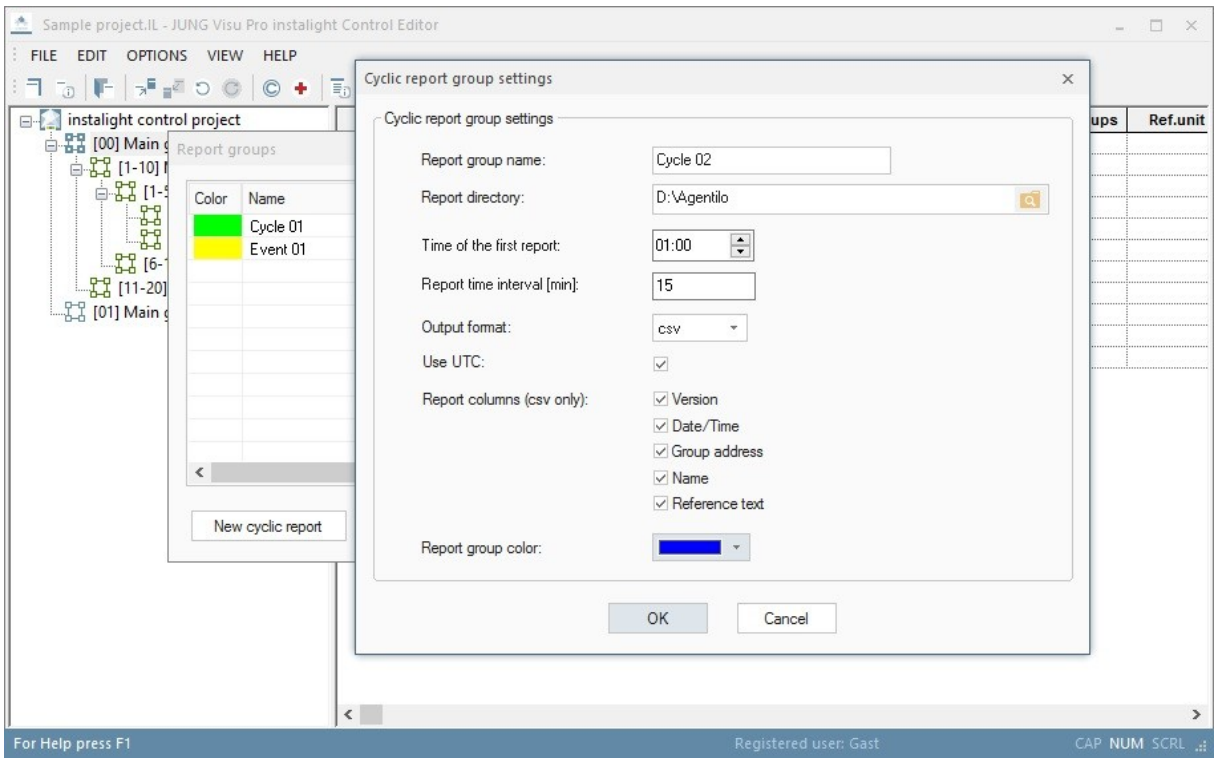
The report groups divide into: ***cyclic reports*** and ***event reports***.

Cyclic reports record from a certain time, in defined periods, the state of the data points within the group.

Event reports check after a preset time period if the value within the group changed and record this value in an event report. It can additionally be determined after what a time period of constant value the value should be recorded as an event.

Click under ***Options*** at ***Edit report groups*** to open the dialog ***Report groups***.

Create report groups:



Click either at the button ***New cycl. report*** or ***New event report*** to open the dialog ***Report group – settings***. Enter the group settings and confirm with ***OK***. The new start group is now listed at the dialog ***Report groups***.

Name of report group: Name of the group with which the group will be listed.

Report catalog: The path to the folder that contains the saved reports.

Time of first request (only at cyclic report): Time between 00:00 and 23:59.

Report interval [min]: The time in minutes after which a new report is written. Oriented at the time of the first report. Between 1 and 86400 minutes.

Minimum time necessary of a value to stay constant [sec] (only at event report): Between 1 and 60 seconds.

Output format: The reports are written either into a xml or a csv file.

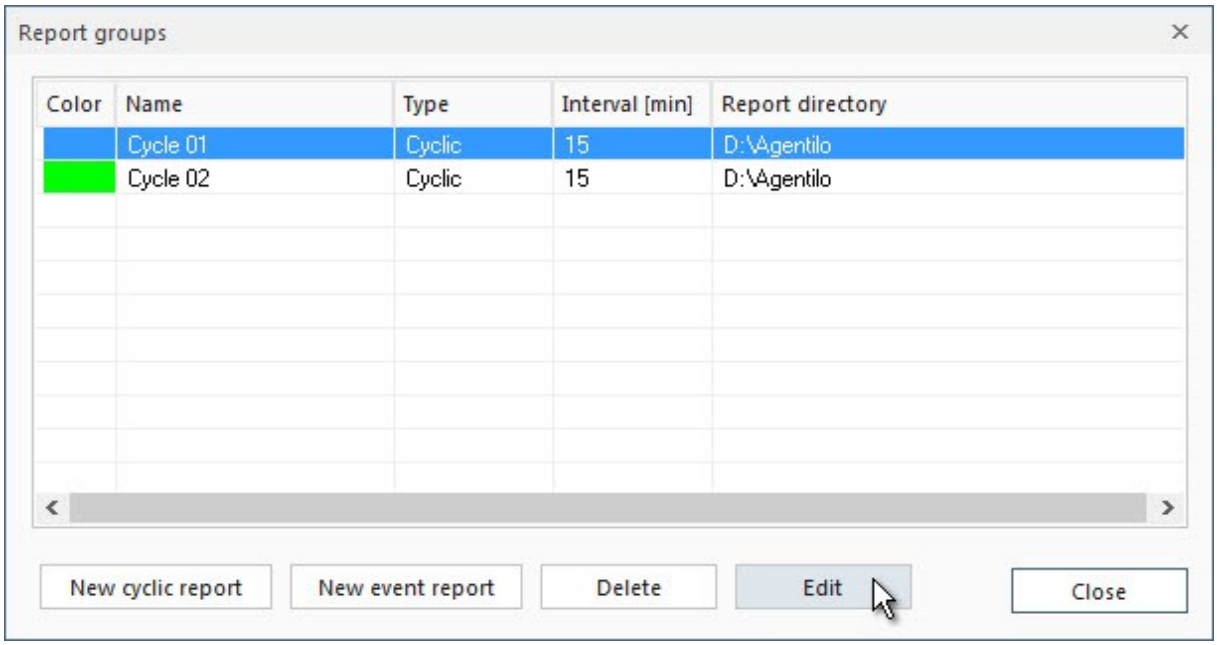
Use UTC: Activates or deactivates the Unit-Time-Code for writing the report.

Report columns (only csv): Defines which additional information should be written into the report.

Color of report group: For a better discrimination of the different groups, a color can be set.

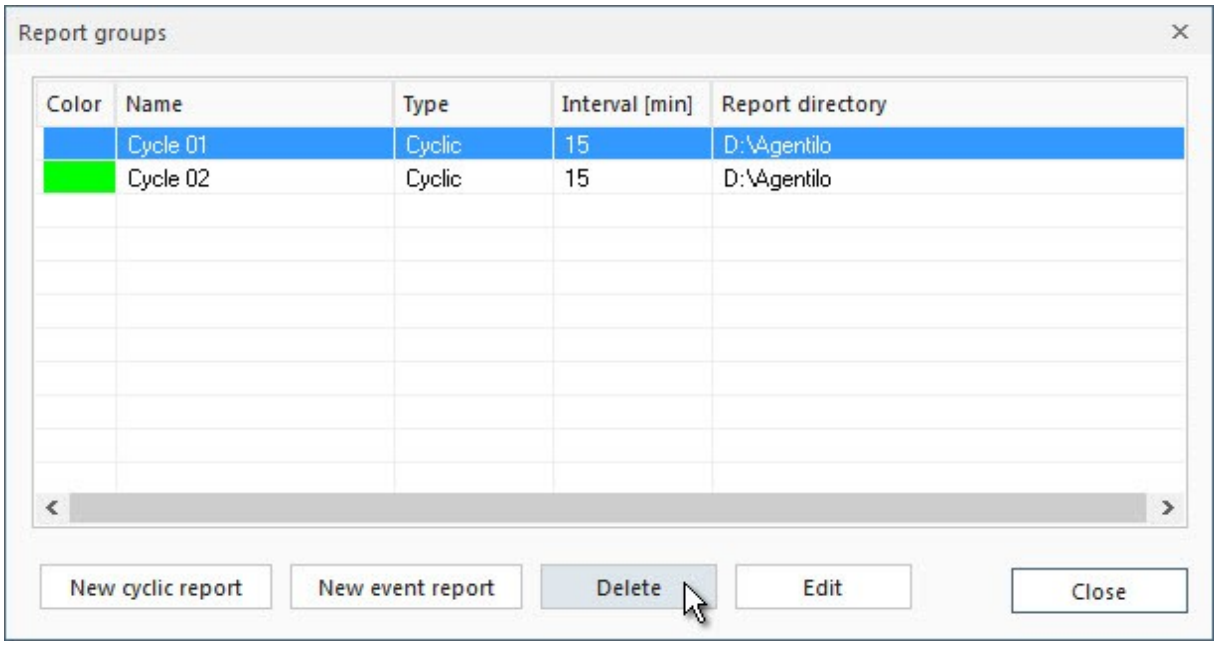
Edit report group:

To edit a report group, select the report group and click at ***Edit***. The dialog ***Report group settings*** opens.



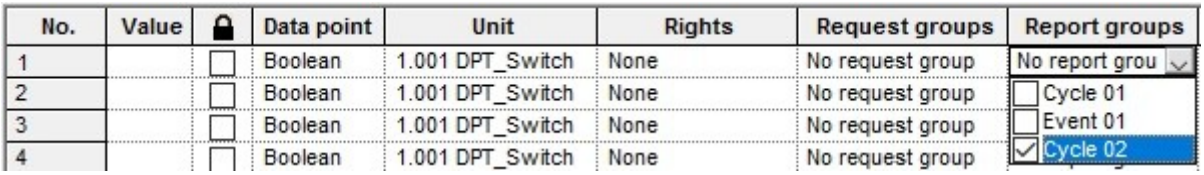
Edit the group settings and confirm the changes with ***OK***.

Delete report group:



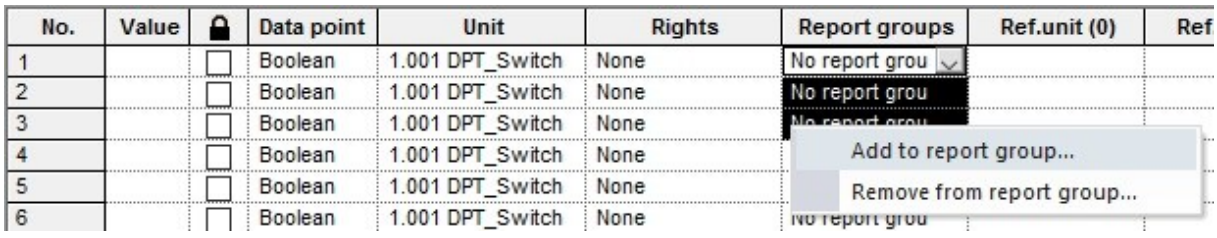
To delete a report group, select the report group and click at ***Delete***. Afterwards, click at ***Finish*** to close the dialog ***Report groups***.

Assign report group:

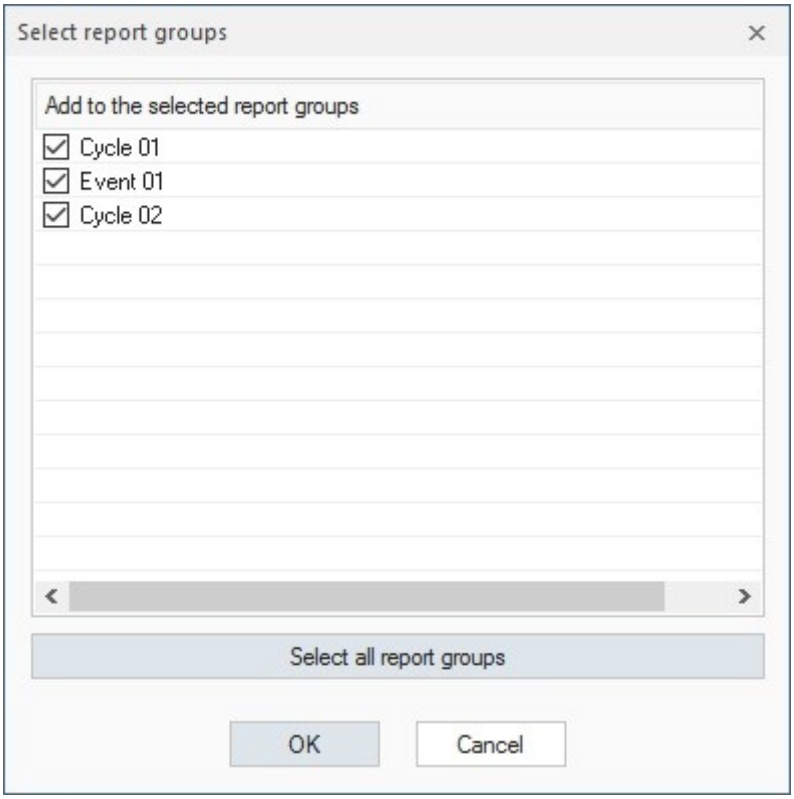


To assign a report group to a group address, click at the column ***Report groups*** to select a report group.

Or: To assign a selection of group addresses to a report group, select the first group address in the column ***Report groups*** and draw the mouse over the beneath lying lines. Right-click at the selected elements to open the context menu and click at ***Add to report group***.



The dialog ***Report group*** opens.



Set a checkmark at the desired report groups and confirm the selection with ***OK***.

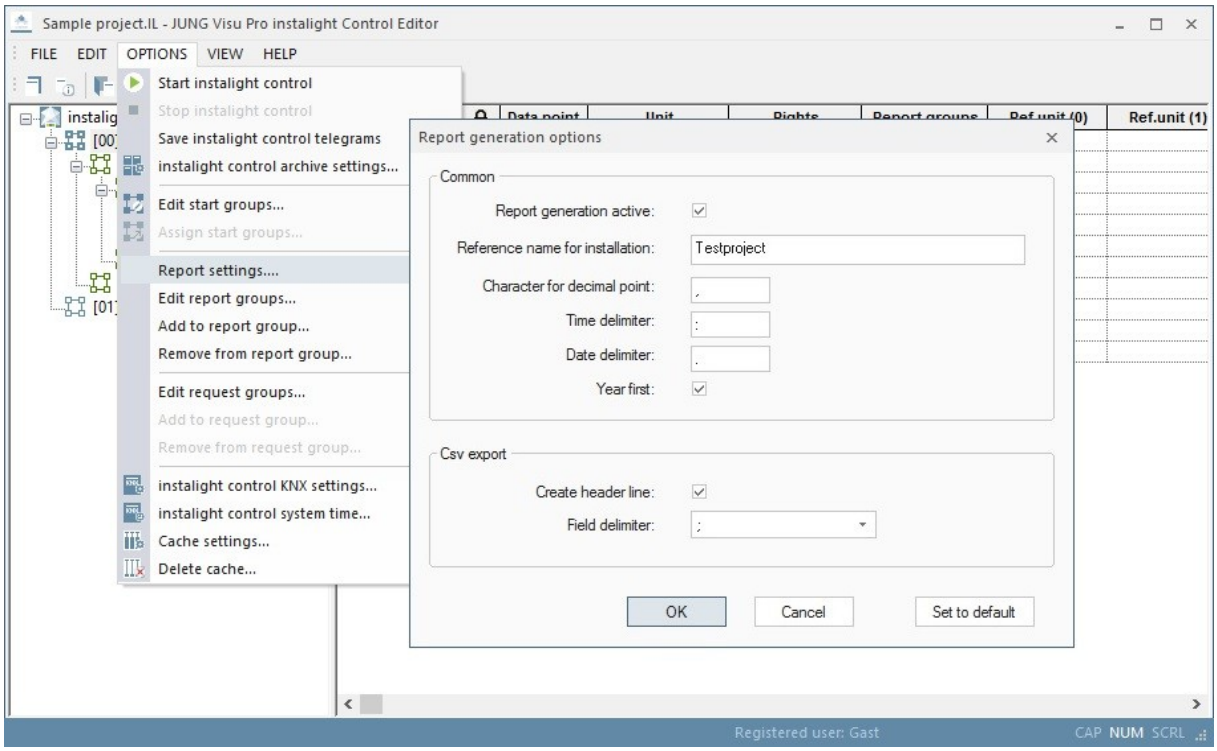
No.	Value	🔒	Data point	Unit	Rights	Report groups
1		☐	Boolean	1.001 DPT_Switch	None	3 report group
2		☐	Boolean	1.001 DPT_Switch	None	3 report group
3		☐	Boolean	1.001 DPT_Switch	None	Cycle 02
4		☐	Boolean	1.001 DPT_Switch	None	Cycle 01
5		☐	Boolean	1.001 DPT_Switch	None	Event 01
6		☐	Boolean	1.001 DPT_Switch	None	2 report group

For every group address is the report group now visible or, if a group address has multiple report groups, the number of report groups.

Delete report groups:
Deleting report groups is analog to assigning report groups.

4.6.1 Report formatting

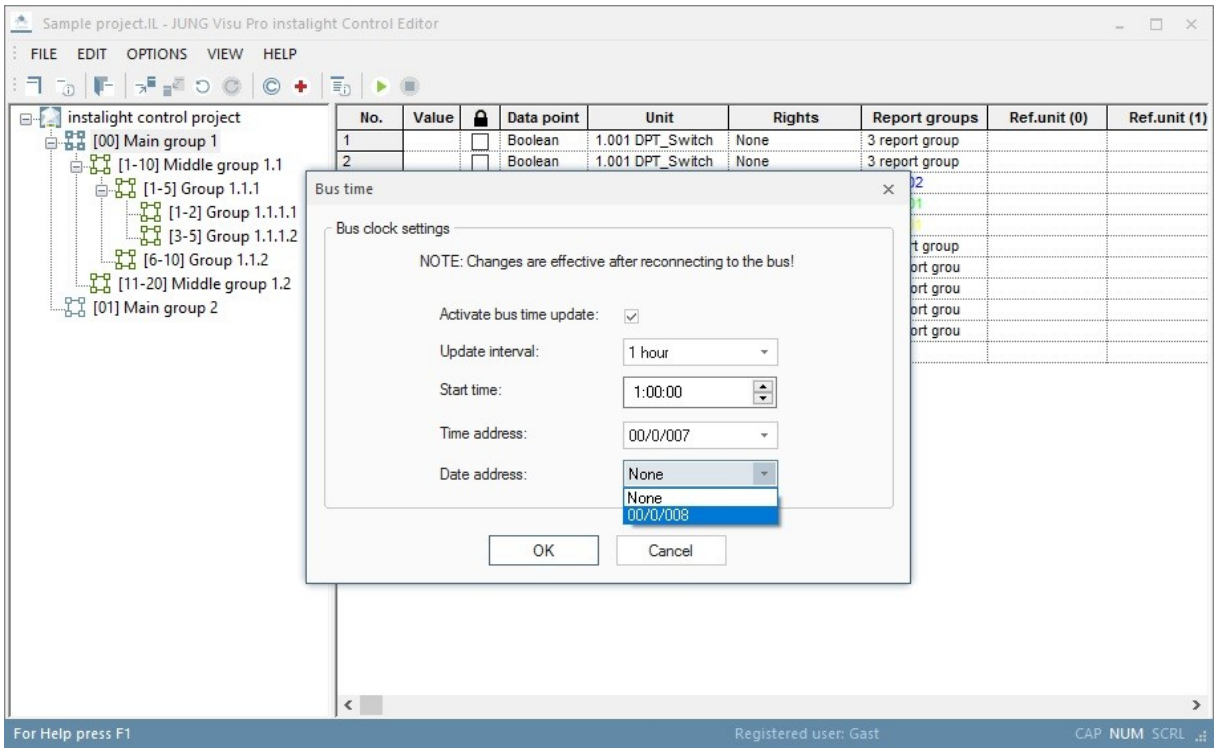
If the produced csv format is not of use for further editing, it is possible to change the formatting of the reports. Click under ***Options*** at ***Report settings*** to open the dialog ***Options for the report creation***.



Change the settings and confirm with ***OK***.

4.7 instalight control KNX System Time

To activate the actualization of the bus time, choose the menu item ***Options*** and click ***instalight control KNX system time***. The dialog ***Bus time*** opens.



To **actualize** the bus time **automatically**, activate the check box ***Activate bus time update***.

Set the update interval and the start time and choose the write addresses for date and time. Press ***OK*** to commit the changes and to close the dialog.

4.8 Application of KNX Data Types

The KNX data point types specify the meaning of transferred bus data. There are different interpretations of the raw data depending on their data type.

The data packet **cf c7**, received by the instalight editor, can have completely different meanings:

-291.84,	interpreted as 2 Octet Float Value
53191,	interpreted as 2 Octet Unsigned Value
–12345,	interpreted as 2 Octet Signed Value

Attention: *The correct assignment of KNX data point types to group addresses is a fundamental precondition for all applications which use the instalight editor as process connection!*

The following table gives a short introduction to the meaning of KNX data point types:

Boolean	**Unit** **1.001 –1.022** **DPT**	**1 bit** data length, formerly EIS1. This data type is used to transmit switch states (e.g. 0/1, on/off, open/closed, up/down, …). This type is particularly used to switch devices on or off or to receive device states (lamps, dimmers, shutters, relay, …).
1-Bit Controlled	**Unit** **2.001 – 2.012** **DPT**	**2 bit** data length, formerly EIS8. This data type is used for priority-controlled switch operations.
3-Bit Controlled	**Unit** **3.007 – 3.008** **DPT**	**4 bit** data length, formerly EIS2, subtype control. This data type is used to increase or decrease the set value in steps or stop the movement (e.g. to control a dimmer).
Character Set	**Unit** **4.001- 4.002** **DPT**	**1 byte** data length, formerly EIS13. This data type is used to transfer a character over the bus. ***Attention:*** *The visualization does not yet support this data type.*

8 Bit Unsigned Value	**Unit 5.001, 5.003, 5.004, 5.010**	**1 byte** data length, formerly EIS6 or EIS14. This data type is used to transfer 8 bit unsigned integer values (e.g. counter values).

> ***Attention:*** *Normally the range of the 8 bit value is from 0 to 255, but the interpretation can differ depending on the subtype selection. If the subtype* scaling *is selected, the value is interpreted as a percent value in the range from 0 to 100%. This subtype can be used to set the values of dimming actuators. If the subtype* wind direction *is selected, the value is interpreted as an angle in the range from 0 to 360°.*

Subtype 5.001 DPT_scaling	The value is interpreted as a percent value in the range from 0 … 100%.
Subtype 5.003 DPT_angle	The value is interpreted as an angle in the range from 0 … 360°.
Subtype 5.010 DPT_Value_Ucount	The value is interpreted in the range from 0 … 255, e.g. as counter value.

8 Bit Signed Value	**Unit 6.010 DPT**	**1 byte** data length, formerly EIS14, used to transfer an 8 bit signed integer value in the range from –128 to +127.
2 Octet Unsigned Value	**Unit 7.001, 7.010, 7.600 DPT**	**2 byte** data length, formerly EIS10, used to transfer a 2 byte unsigned integer value (e.g. a counter value).
2 Octet Signed Value	**Unit 8.001 DPT**	**2 byte** data length, formerly EIS10 signed, used to transfer a 2 byte signed integer value in the range from – 32768 to +32767.
2 Octet Float Value	**Unit 9.001-9.006, 9.010-9.011. 9.020-9.021 DPT**	**2 byte** data length, formerly EIS5, used to transfer a 2 byte analog value (e.g. values from a temperature sensor).
Time	**Unit 10.001 DPT**	**3 byte** data length, formerly EIS3, used to transfer a time information (e.g. from a KNX synchronization clock).
Date	**Unit 11.001 DPT**	**3 byte** data length, formerly EIS4, used to transfer a date value.

4 Octet Unsigned Value	**Unit 12.001**	**4 byte** data length, formerly EIS11, used to transfer a 4 byte unsigned integer value (e.g. a counter value).
4 Octet Signed Value	**Unit 13.001**	**4 byte** data length, formerly EIS11 signed, used to transfer a 4 byte signed integer value in the range from -2147483648 to +2147483647.
4 Octet Float Value	**Unit 14.001-14.079 DPT**	**4 byte** data length, formerly EIS9, used to transfer a 4 byte high precision analog value according to IEEE754 standard.
Access		**4 byte** data length, formerly EIS9, used for access control applications.

> ***Attention:***
> *The visualization does not yet support this data type.*

String	**Unit 16.000-16.001 DPT**	**14 byte** data length, formerly EIS15, used to transfer strings.
UTF-8 String	**Unit 28.001 DPT**	**14 byte** data length, used to transfer strings in UTF-8 format.
Colour RGBW	**Unit 251.600 DPT**	**6 byte** data length, used to transfer colors in RGBW format.

4.9 ETS Import of Group Addresses

In most cases it is more convenient to import KNX group addresses from ETS than to create them manually. The instalight editor is able to import KNX group addresses from **ETS2** versions 1.1, 1.2 and 1.3.

There are two methods:

a) Import the data from an ETS print report which has been redirected to a file. This method applies to **ETS2**, versions 1.1, 1.2 and 1.3.
b) Import the data from a file which has been created with the function OPC export. This method applies to **ETS2**, Version 1.3 and **ETS3**.

Data from **ETS4** can be exported to a **project export** file which can be imported in the instalight editor.

> ***Important:*** *The instalight editor does not require ETS to be installed at the same PC. It is sufficient to copy the ETS output files to the PC.*

Please consider the following notes when importing group addresses from ETS:

- Assign **unique names** to the group addresses in the ETS project. It is important to assign meaningful names to the group addresses in order to facilitate the identification of addresses and plain text messages in the instalight editor and the visualization.

- Make sure to set the **data point types** already in the ETS project. The instalight editor is able to import the data point types from the ETS print file. If you have configured the data point types correctly in the ETS project, you do not have to enter them again in the instalight editor. This facilitates in particular the updated import of addresses after changes have been carried out in the ETS project.

- Edit the imported group addresses after the import to the instalight editor:

  → Check whether the **data point types** are set correctly;

  → Set the **read and write rights**. The read and write rights for the instalight editor (and for programs using the instalight editor) **cannot** be imported from ETS. They must be set manually.

4.9.1 Create ETS Group Addresses Print Report

Create ETS export file in ETS:

1. Open the ETS project in ***Project Design***.

2. Open the group address browser (Menu: ***Design***, ***Group Addresses***).

3. Choose the menu item ***Project*** > ***Page Setup*** and click ***Layout***. Click in the layout dialog ***reset the selection of Title page, Header, Footer***.

4. Mark the **root node** in the group address browser.

5. Choose the menu item ***Project*** and click ***Print***. In the print dialog click ***Print to File*** and click the button ***OK*** to start the export.

6. A print dialog opens. Choose the target directory, target file name, and select the extension ***.txt***.

7. To import the output file into the instalight editor, choose the menu item ***File*** and click ***Import ETS project***.

> ***Important:*** *With ETS the installed printer driver must support the function* **printing in a file**. *As a remedy install the WINDOWS standard printer driver.*

4.9.2 Create OPC Export File

The instalight editor can import group addresses from a file which has been created from ETS2 version 1.3, ETS 3 and ETS4 via the function ***OPC Export***.

Create OPC export file in ETS2 version 1.3:

1. Start the function ***OPC Export*** in ETS.

2. The ***OPC Export*** dialog opens. Select the ETS project and specify a target directory and a file name for the OPC project file.

3. Click the button ***Export*** to create the file in the specified directory. The default extension of the OPC project file is ***.esf***.

4. To import the file in the instalight editor, choose the menu item ***File*** and click ***Import ETS project***.

Create OPC export file in ETS3:

1. Choose the menu item ***Extras*** and click ***Options***. Select the tab ***Presentation***. Select under ***Browser*** the check box ***Two Level Group Addresses***.

2. Choose the menu item ***File*** and click ***Extract Data (e.g. OPC)***. The dialog ***Foreign Format*** opens.

3. Click ***Export to OPC Server***. Specify the target directory and a file name for the OPC project file. Then click ***Export***. The default extension of the OPC project file is ***.esf***.

4. To import the file into the instalight editor, choose the menu item ***File*** and click ***Import ETS project***.

5. To toggle between the 2-level, 3-level and the free address structure, choose the menu item ***Edit*** and click the desired address structure.

Create OPC export file in ETS4:

1. Switch in ETS4 to the project overview and select the desired project. The button ***Export*** in the toolbar will be activated.

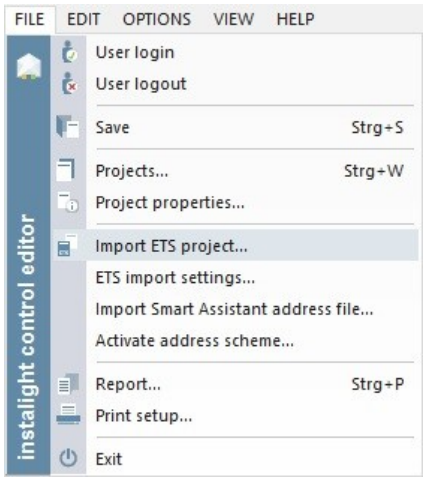
2. Click in the toolbar the button ***Export***.

3. Enter the project name *"Projectname.knxproj"* and click the button ***Save***. The default extension of the file is ***.knxproj***.

4. A dialog indicating the progress of the data transfer opens. After the transfer is completed, confirm the pop up message box ***Operation successful*** by clicking the button ***OK***.

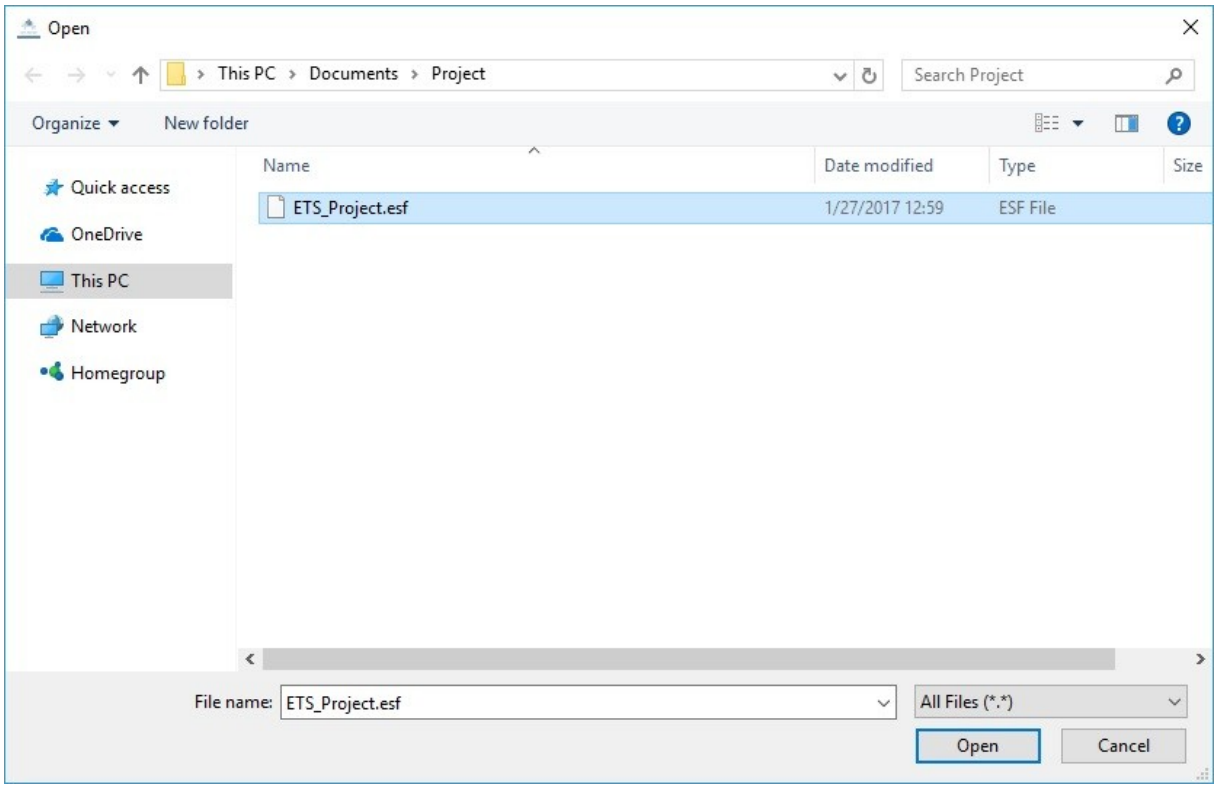
5. The exported ETS4 project is now available in the selected directory and can be imported in the instalight editor.

4.9.3 Import of ETS Export Files into the instalight editor



Import ETS project:

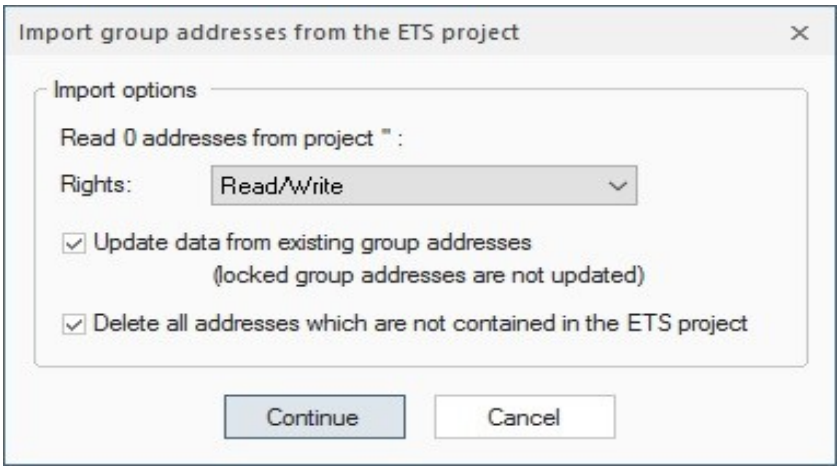
Choose the menu item ***File*** and click ***Import ETS project*** to open the dialog for the selection of the directory.



The redirect print reports usually have the extension ***.txt***, the OPC Export–generated files have the extension ***.esf*** and the ETS4 export files have the extension ***.knxproj***.

Select the file to be imported and click the button ***Open***. A dialog for the import opens. Select the project out of the list from the ETS file and confirm with ***OK***.

A dialog for the import of the group addresses opens.



The dialog shows the number of group addresses in the import file. You can set here the access rights which will be assigned to the imported group addresses by default. Since the rights for read and write access cannot be imported, make sure to check them and, if necessary, correct them after the export. But at this step you may select the default rights which are most convenient for the project.

Press ***Continue*** to load the ETS project into the editor.

Import options:

With the check boxes at the import dialog, the following options can be set:

Update data from existing group addresses:

If this option is set, the data of already existing group addresses in the instalight editor will be actualized.

Note: *Enable this option if you want to transfer the changes carried out in the ETS project to the group addresses already contained in the instalight editor (i.e. change of address names).*

Lock and unlock group addresses:

You may want to prevent a data point from being updated. To **lock a data point** for the update, select the checkbox in the column with the padlock icon.

No.	Address	Designation	Value	🔒	Data point type	Unit	Ri
1	00/0/006	Group 6		☑	Boolean	1.001 DPT_Switch	Read/W
2	00/0/007	Group 7		☐	Boolean	1.001 DPT_Switch	Read

To lock several group addresses, click a group address in the column with the padlock icon, hold down the Ctrl key and extend the selection on further group addresses. Then right-click and click in the context menu the option ***Lock address***.

To lock a range of group addresses, click the first group address in the column with the padlock icon, hold down the shift key and click the last group address to be locked. Alternatively, click with the left mouse button the first address to be locked and move the mouse cursor to the last address to be locked. Then right-click and click in the context menu the option ***Lock address***.

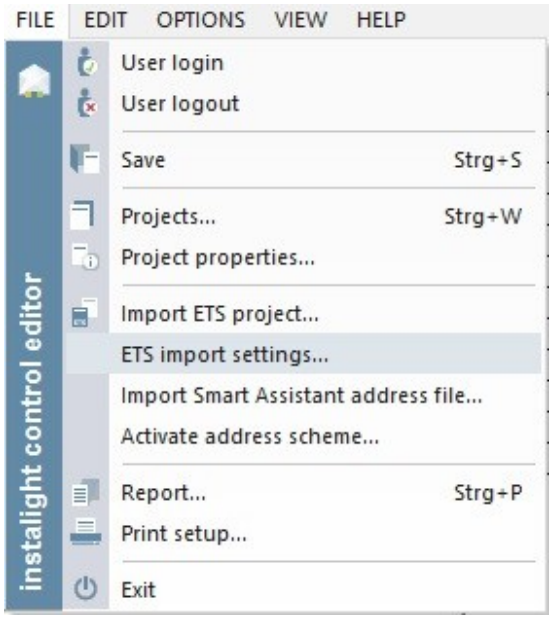
To unlock an address for the update, clear the check box or right-click and click in the context menu the option ***Unlock address***.

> ***Note:*** *If you check the option* ***Delete all addresses which are not contained in the ETS project****, the locked group addresses will also be deleted if they are not contained in the ETS project.*

Delete all addresses, which are not contained in the ETS project:

If this option is enabled, all addresses will be deleted which are available in the instalight editor but not in the ETS project.

4.9.4 ETS-Import Settings...

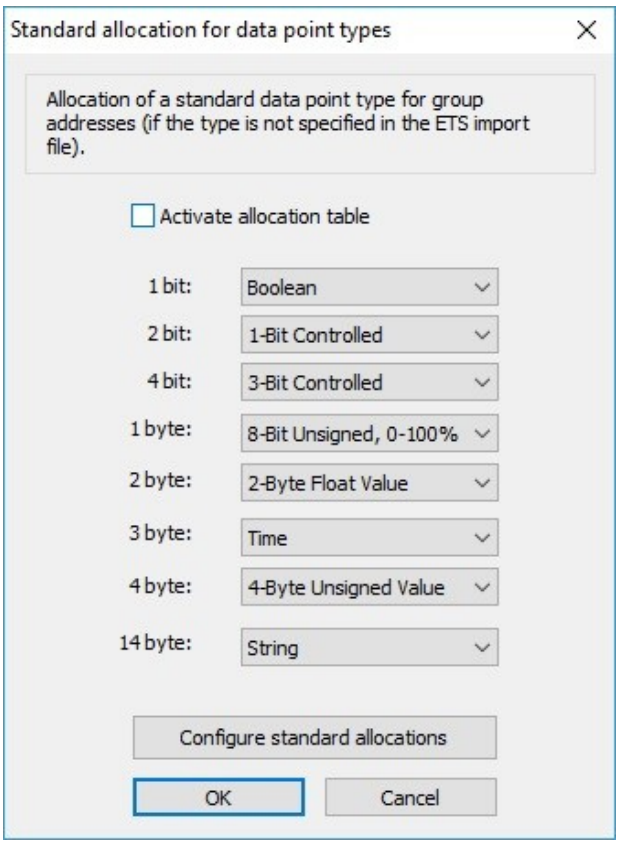


ETS-Import Settings:

Choose the menu item ***File*** and click ***ETS-Import Settings...*** to open the dialog for the standard type declaration for data points.

In this dialog it is possible to activate the declaration table. The decalration table sets standard types for every data point, which has no type specified after the importation of an ETS file.

You can specify a standard data point type foreach available data point length.



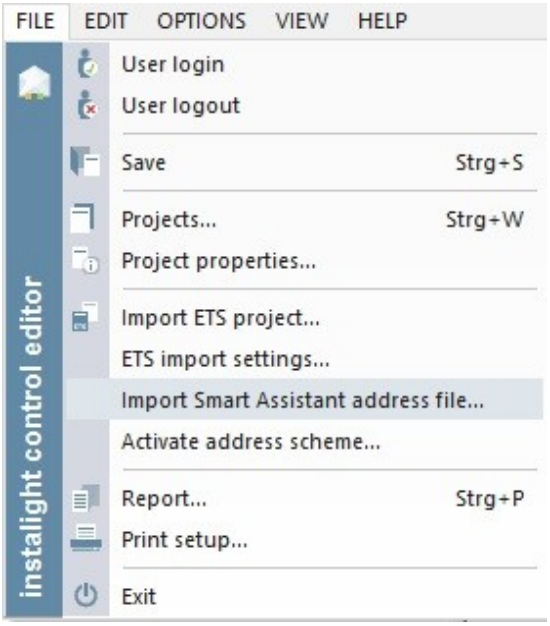
Following data point types are the default values of the declaration table:

1 Bit	Boolean
2 Bit	1-Bit Controlled
4 Bit	3-Bit Controlled
1 Byte	8-Bit Unsigned, 0-100%
2 Byte	2-Byte Float Value
3 Byte	Time
4 Byte	4-Byte Unsigned Value
14 Byte	String

These standard declarations can be restored by clicking the ***Reset to standard declarations*** button.

4.10 Import Of Smart Assistant Address Files

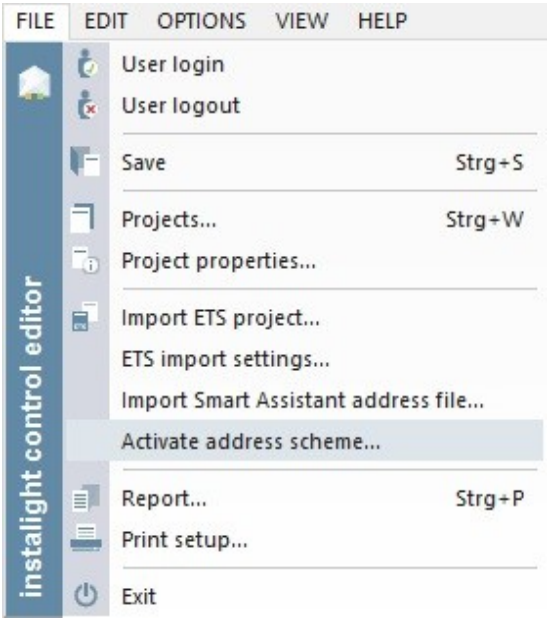
It is possible to import address files, generated by the JUNG Visu Pro Smart Assistant, into the KNX editor.



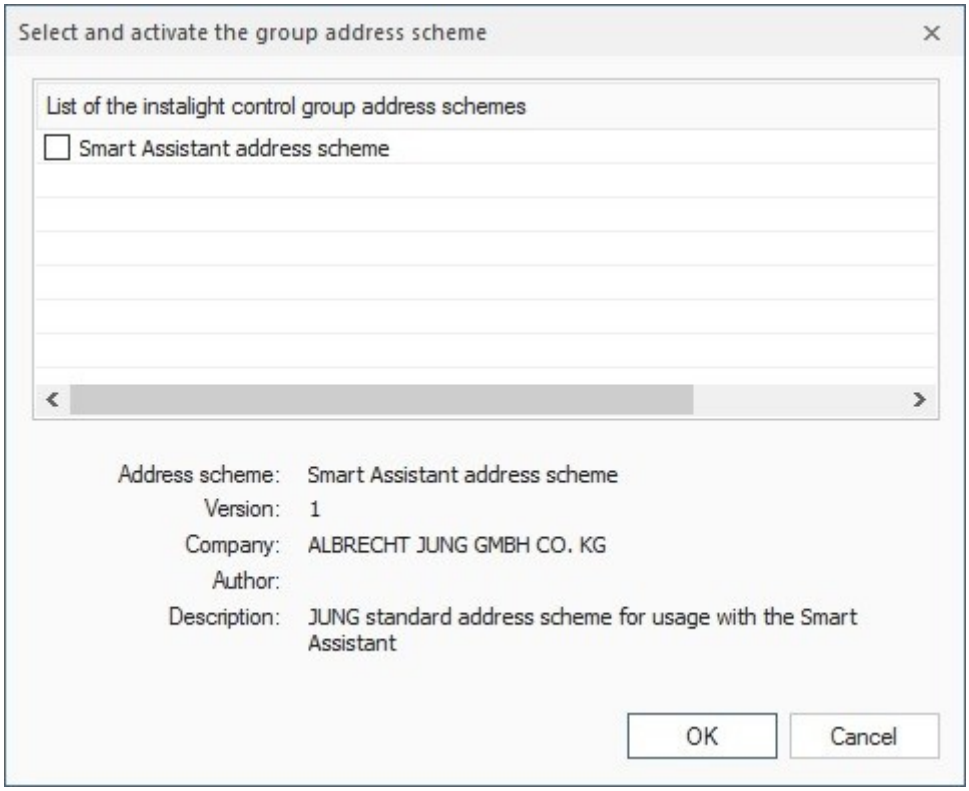
Import of a Smart Assistant Address File:

Choose the menu item ***File*** and click ***Import of a Smart Assistant Address File...*** to open the dialog for the selection of the address file.

4.11 Activate Address scheme

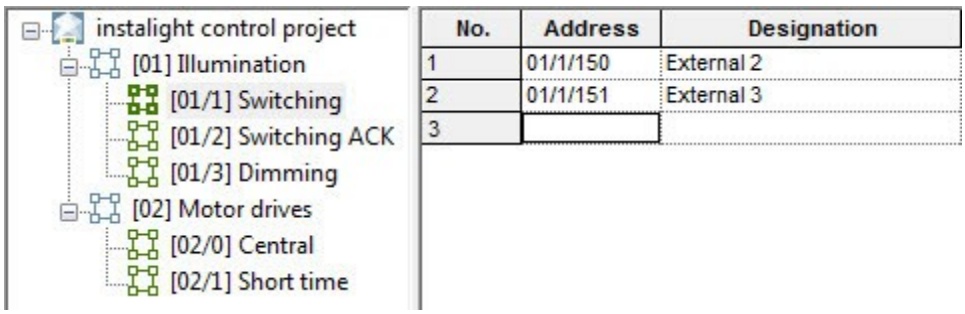


Choose the menu item ***File*** and click ***Activate Address scheme...*** to open the dialog for the selection of the address scheme.



This dialog offers schemes, which can be activated to define specific labels for main and middle groups or to set a standard datapoint type for datapoints in specific middle groups. At the moment the only available scheme is the JUNG Visu Pro Smart Assistant scheme.

The JUNG Visu Pro Smart Assistant scheme offers the same structure for your KNX project, which is used when creating a Smart Assistant project. There is no need to define the labels on your own.



4.12 instalight control KNX Settings

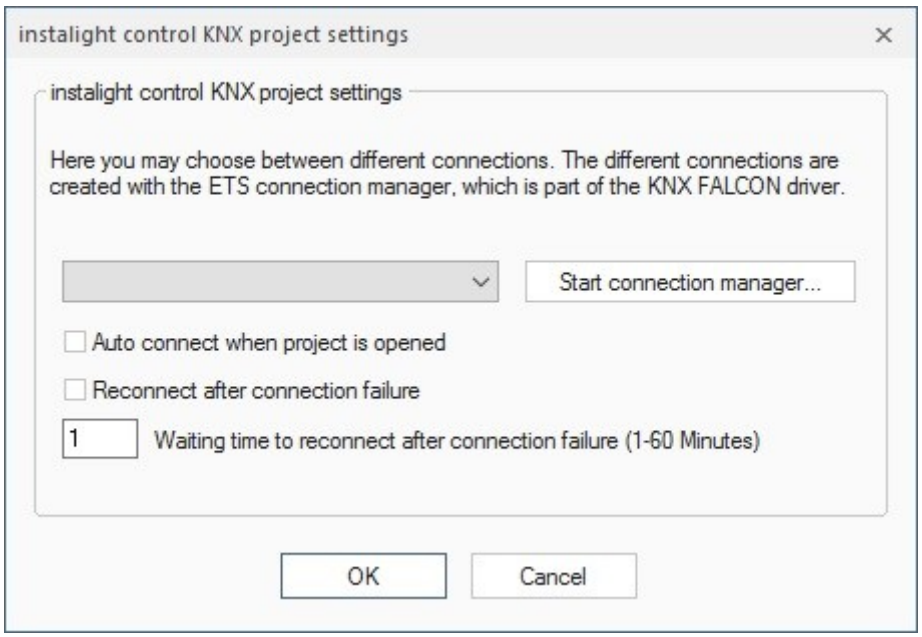
Connect PC to KNX:

The connection to KNX is realized via the FALCON driver (of the KNX Association) which is contained in the instalight editor. The Falcon driver supports the following interfaces: RS 232 (PEI10 and PEI16), KNX-USB, EIBlib/IP and KNXnet/IP. The PC must provide one of the following interfaces: RS232, USB, IP.

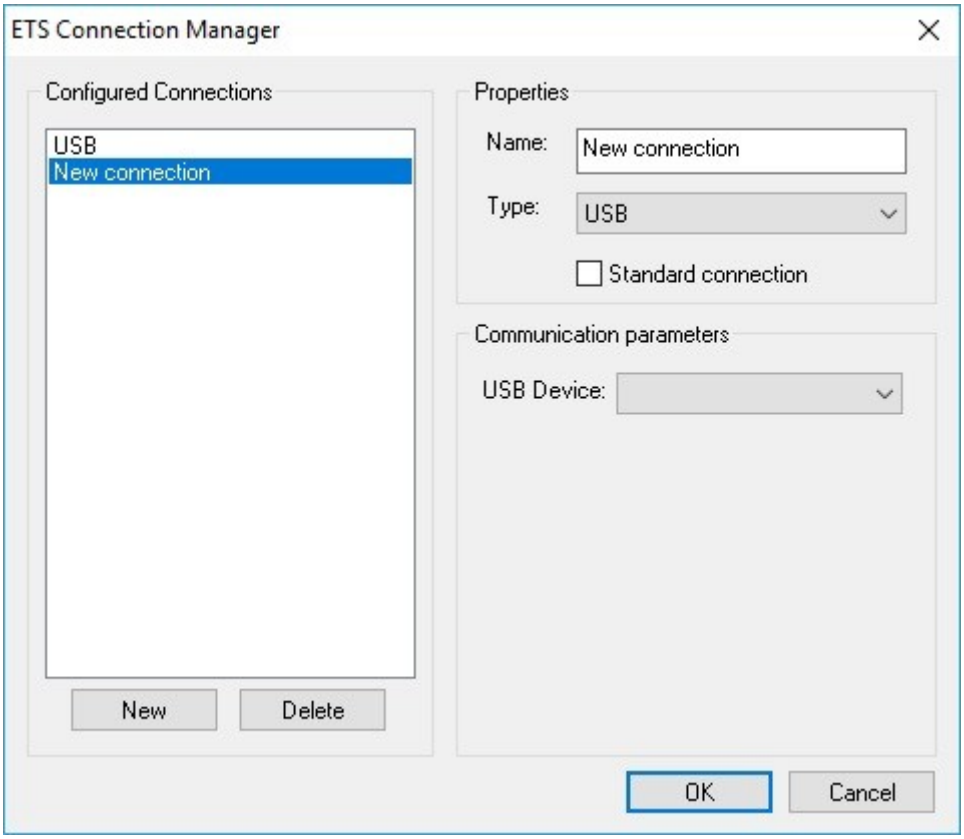
Set KNX parameters:

In the instalight editor the Falcon driver is configured in such a way that the connection can be established via one of the interfaces. The connection to the serial interface of the BCU is possible via the handshake protocol (PEI16) or FT1.2 protocol (PEI10).

Choose the menu item ***Options*** and click ***instalight control KNX settings***. The dialog ***instalight control KNX project settings*** opens.

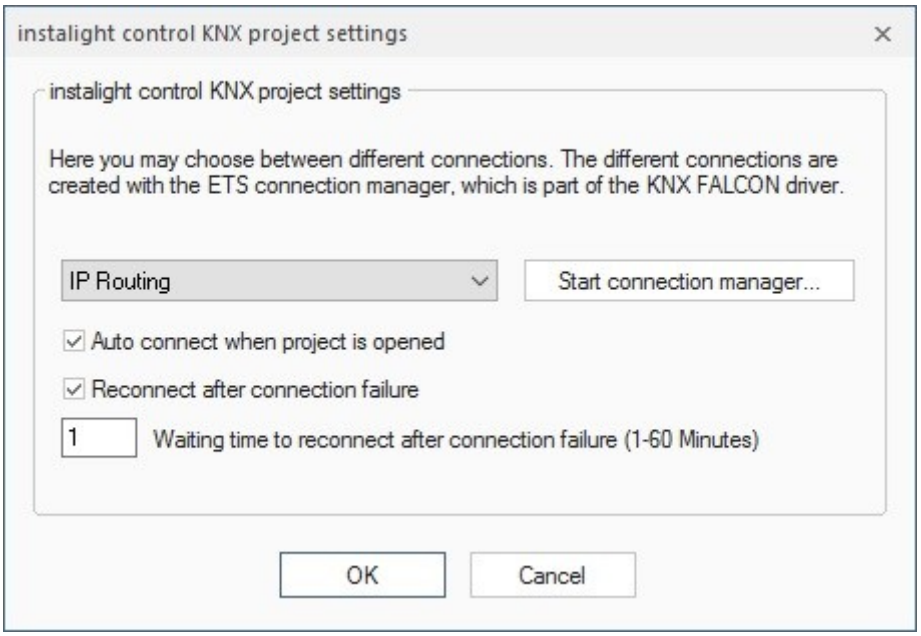


Here you can specify which of the KNX connections configured for the PC should be used to establish the access to the bus. The different connections are created with the help of the ETS connection manager which is part of the FALCON driver. Click the button ***Start connection manager*** to start the ETS connection manager dialog.



Click the button ***New*** to create a new connection. In the example below a standard connection for the USB interface has been created. The name can be edited. The connection type and the USB device can be selected.

Click ***OK*** to commit the entries.



Auto connect when project is opened:

The connection to the bus can be established automatically right after the start of the instalight control project. To enable this option, select the check box ***Auto connect when project is opened*** in the dialog ***instalight control KNX project settings***. Thus, you do not have to establish the connection manually.

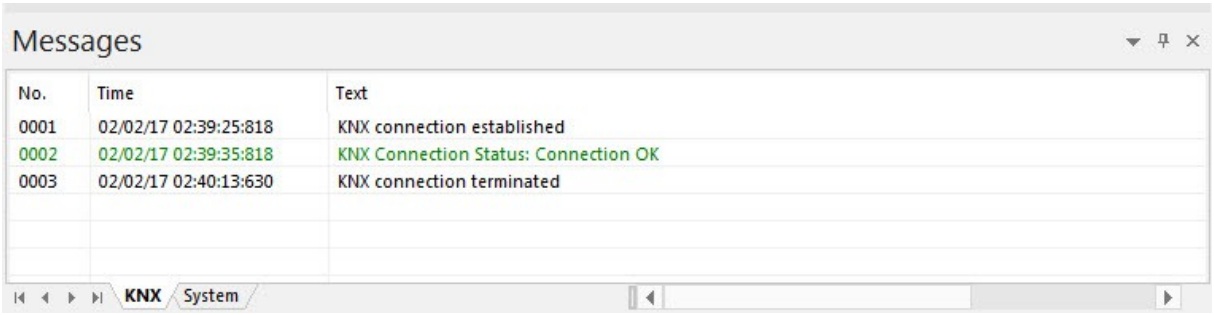
Reconnect after connection failure:

If this option is enabled, the system reconnects automatically to the bus in case of a connection failure.

4.13 Connect to instalight control KNX Bus

To start the connection to the instalight control KNX bus, choose the menu item ***Options*** and click ***Start instalight control***. Alternatively, click on the symbol in the toolbar.

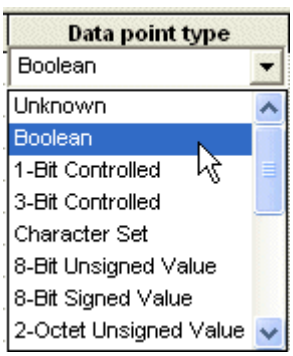
To interrupt the connection to the bus, choose the menu item ***Options*** and click ***Stop instalight control*** or click the symbol in the toolbar.

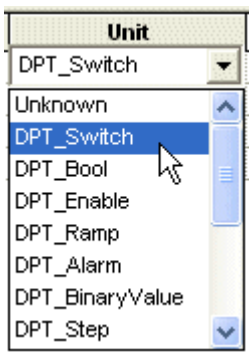


You can see the results of these operations and possibly already received telegrams in the message list.

4.14 Send Telegrams Manually

In the KNX group addresses grid make sure that the columns ***Data point type***, ***Unit***, and ***Rights*** contain the proper value for each KNX group address.





If the KNX group address is properly defined and the write access is allowed, a value can be entered in the column ***Value***. Alternatively, right-click in the value column and select the desired value in the context menu (picture below).

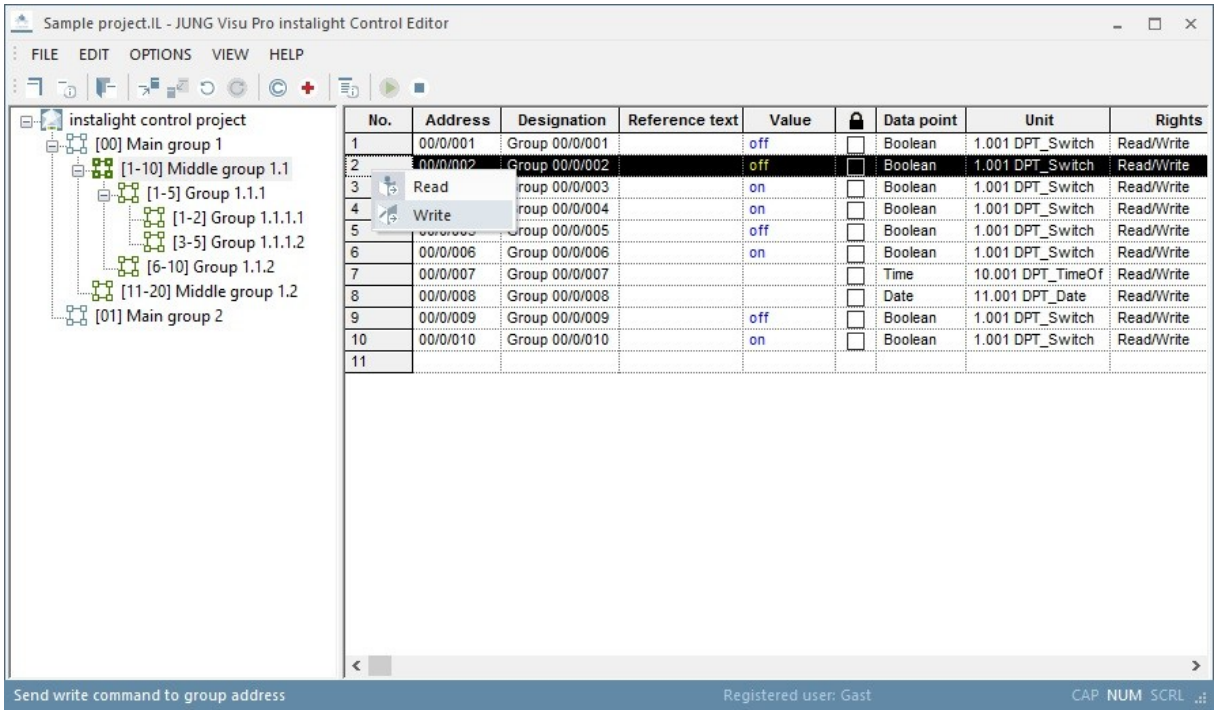
No.	Address	Designation	Value	🔒	Data point type	Unit	Rights
1	00/0/001	Date	1/13/2017	☐	Date	11.001 DPT_Date	Read/Write
2	00/0/002	Time	fri, 12:00:00	☐	Time	10.001 DPT_TimeO	Read/Write
3	00/0/003	Temperature	0.00 °C	☐	2-Byte Float Valu	9.001 DPT_Value_	Read/Write
4	00/0/004	Switch	off	☐	Boolean	1.001 DPT_Switch	Read/Write
5	00/0/005	Switch	off		Boolean	1.001 DPT_Switch	Read/Write
6	00/0/006	Lights	off		Boolean	1.001 DPT_Switch	Read/Write
7	00/0/007	Blinds	off		Boolean	1.001 DPT_Switch	Read/Write
8							

0 off
1 on

For binary values (type Boolean) use ***0*** for ***off*** and ***1*** for ***on***. For other types just enter the value. The instalight editor will perform the required data conversions.

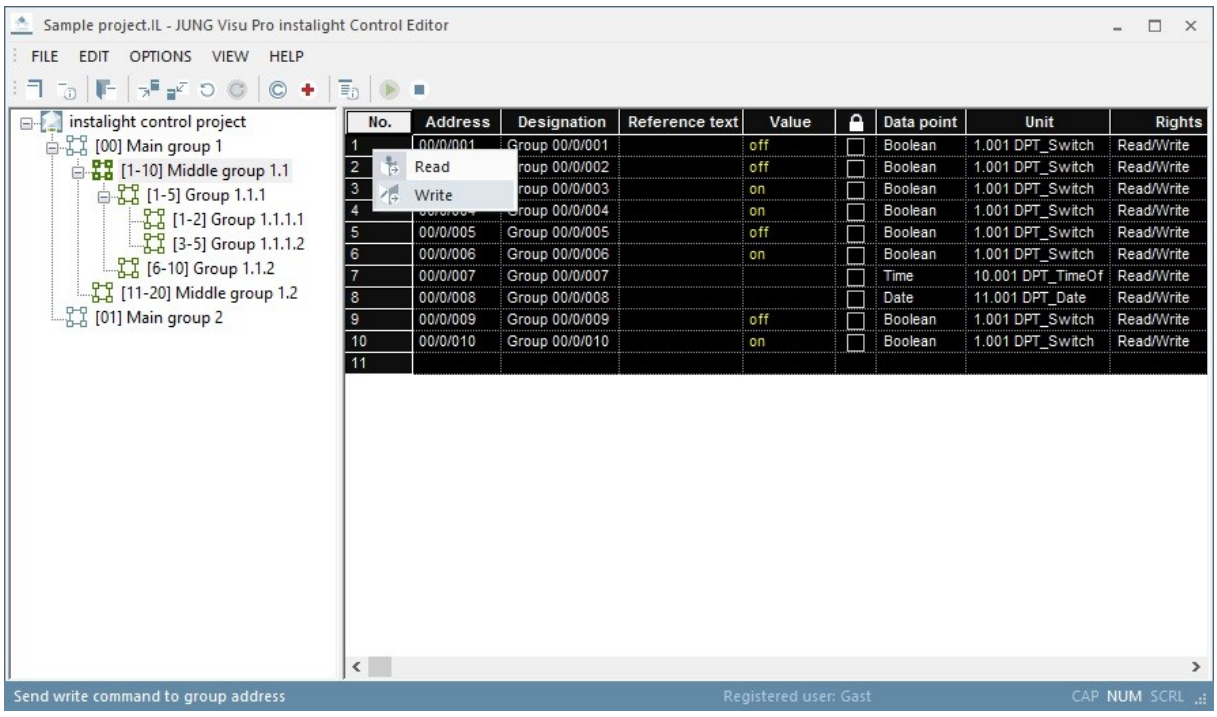
You can enter an analog value, for example **25.65** for the data point type 2 Octet Float. When the value is to be sent, the content of this column is read, converted into an instalight control KNX telegram on the basis of the specified data point type and sent via the bus.

To send the value via the bus, select the entire row of the group address with a left click on the ***No.*** column in the row of the group address. Then right-click and select the desired command, ***Read*** for sending or ***Write*** for querying the value.



After the command has been performed, the telegrams are shown in the message list.

To send several commands at the same time, click the ***No.*** column. Then perform a right-click and select the desired command in the context menu.

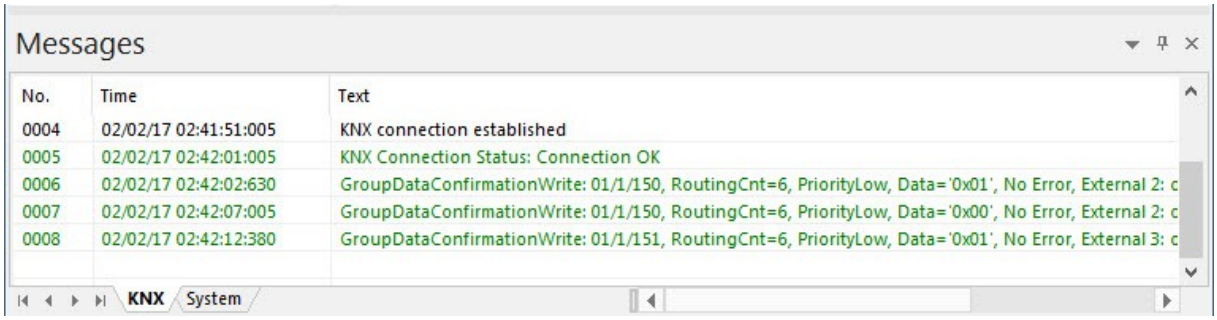


4.15 Message List

The message list shows all instalight control KNX telegrams sent and received. When sufficient information regarding KNX data point type and unit has been supplied, the telegrams will be displayed in plain text.

To toggle the visibility of the message list, choose the menu item ***View*** and click ***Messages***. Alternatively, click the symbol in the toolbar.

Choose the menu item ***View*** and click ***Show last message*** to specify that the message list scrolls automatically to the last message.



Meaning of the colors in the message list:

Green: Sent and acknowledged telegrams.

Blue: Received telegrams.

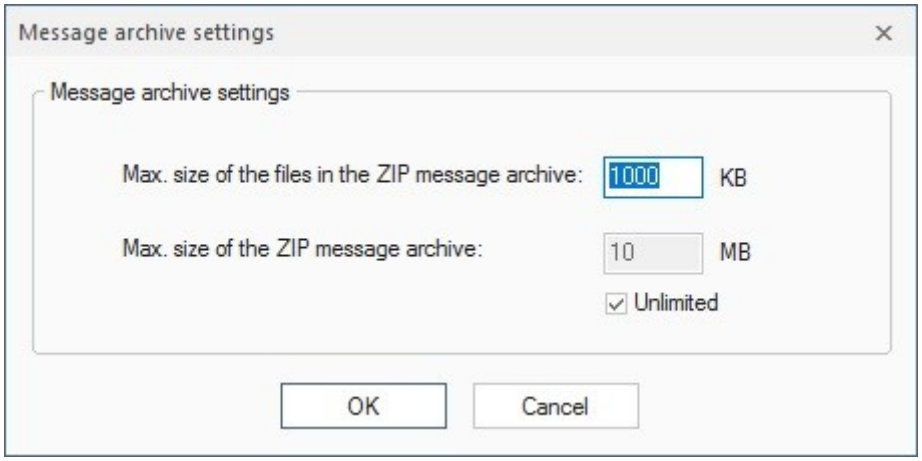
Red: Status or error messages.

Grey: Read requests.

4.16 Logging instalight control KNX Messages into Archives

Messages displayed in the message list can be automatically saved in a text file. Choose the menu item ***Options*** and click ***Save instalight control KNX telegrams*** to activate or deactivate the automatic logging.

To configure the settings for the message archive, choose the menu item ***Options*** and click ***instalight control KNX archive settings***. The dialog ***Message archive settings*** opens.



When the telegram log file reaches a size of about 1 Megabyte, it will be closed and saved into the zip archive **EibMessages.zip** at the directory of the instalight control project:

(...\stations\local\projects\<projectname>.IL\EibMessages.zip).

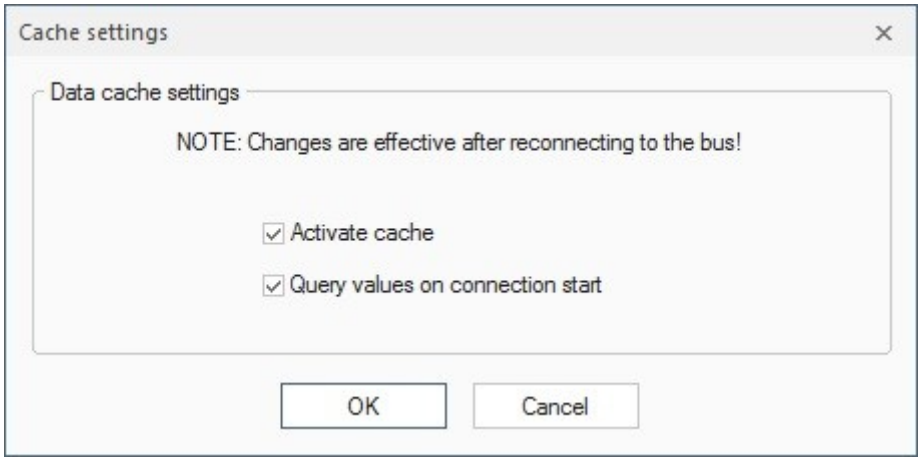
A new log file will be created. The log file will also be closed and saved in the zip archive when the instalight control KNX bus has been disconnected.

4.17 Data Cache Settings

The instalight editor comprises a data cache to optimize access to current values of KNX group addresses. Optionally, the cache can be initialized once by querying the bus, but if the values of KNX group addresses are available in the cache, it will update the cache from listening to the KNX without additional queries.

> ***Note:*** *The cache will always be cleared, when the bus is disconnected.*

Choose the menu item ***Options*** and click ***Cache settings***. The dialog Cache settings opens.



Check boxes within the dialog:

Activate cache: Select this check box to activate the cache.

Query values on connection start: When this check box is selected, the cache will query values for KNX group addresses whenever a connection to the instalight control KNX bus is established.

> ***Attention:*** *Querying values may produce unexpected results when the access rights for the KNX group addresses have not been properly configured.*
>
> ***Therefore check the settings of the access rights before enabling this option.***

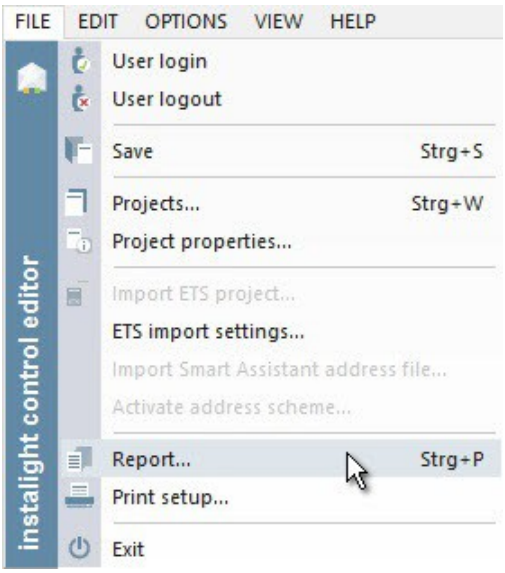
Choose the menu item ***Options*** and click ***Clear cache*** to clear the cache manually.

5 Reports

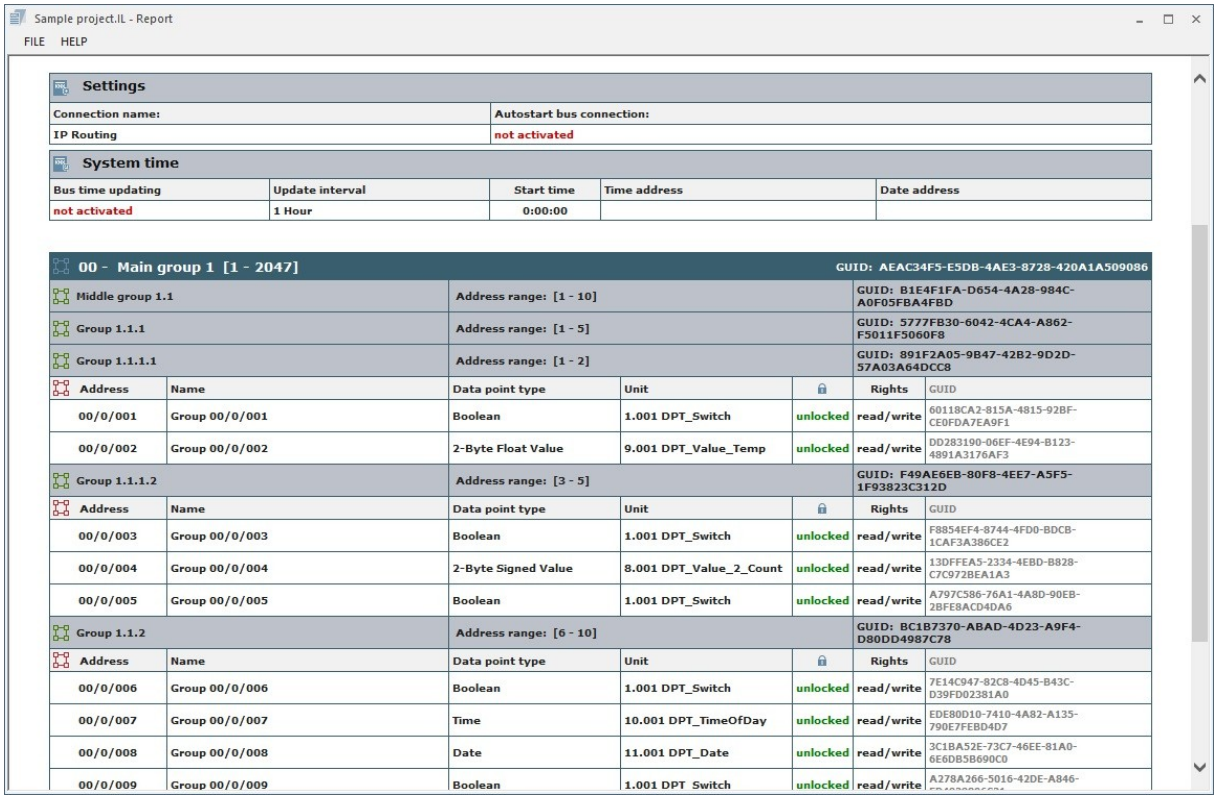
- A report can be created which gives information about the process variables contained in the project and their properties.
- Reports can be printed out (with print preview).

5.1 Create Report

To create a report for the current project, choose the menu item ***File*** and click ***Report***. The dialog ***Create report*** opens. Select the desired options for the report generation and click the button ***Create***.



The data points in the project may be sorted by either names or group addresses.



Sample project.IL - Report

FILE HELP

Settings

Connection name:	Autostart bus connection:
IP Routing	not activated

System time

Bus time updating	Update interval	Start time	Time address	Date address
not activated	1 Hour	0:00:00		

00 - Main group 1 [1 - 2047] GUID: AEAC34F5-E5DB-4AE3-8728-420A1A509086

Middle group 1.1	Address range: [1 - 10]	GUID: B1E4F1FA-D654-4A28-984C-A0F05FBA4FBD
Group 1.1.1	Address range: [1 - 5]	GUID: 5777FB30-6042-4CA4-A862-F5011F5060F8
Group 1.1.1.1	Address range: [1 - 2]	GUID: 891F2A05-9B47-42B2-9D2D-57A03A64DCC8

Address	Name	Data point type	Unit		Rights	GUID
00/0/001	Group 00/0/001	Boolean	1.001 DPT_Switch	unlocked	read/write	60118CA2-815A-4815-92BF-CE0FDA7EA9F1
00/0/002	Group 00/0/002	2-Byte Float Value	9.001 DPT_Value_Temp	unlocked	read/write	DD283190-06EF-4E94-B123-4891A3176AF3

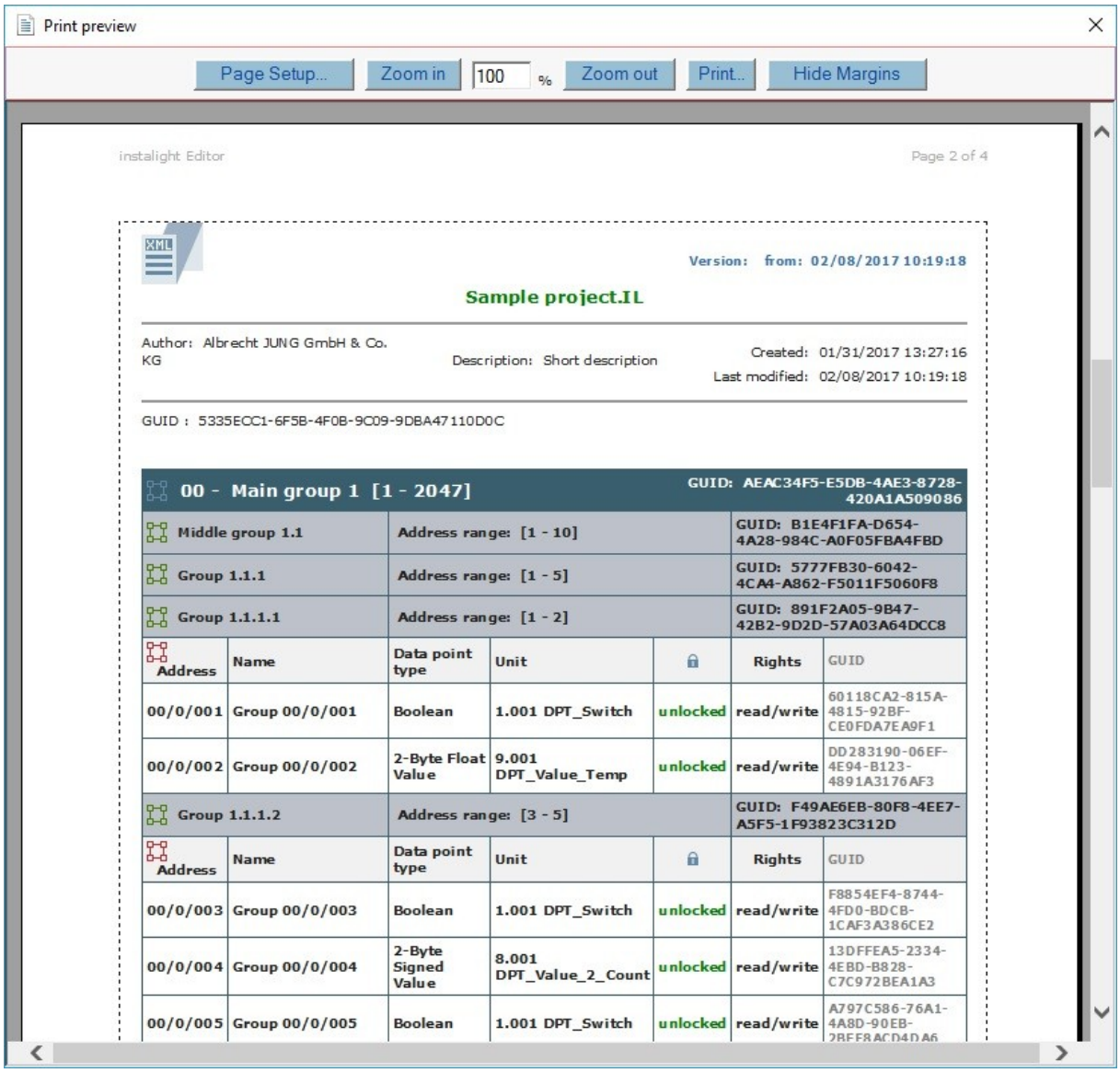
Group 1.1.1.2	Address range: [3 - 5]	GUID: F49AE6EB-80F8-4EE7-A5F5-1F93823C312D

Address	Name	Data point type	Unit		Rights	GUID
00/0/003	Group 00/0/003	Boolean	1.001 DPT_Switch	unlocked	read/write	F8854EF4-8744-4FD0-BDCB-1CAF3A386CE2
00/0/004	Group 00/0/004	2-Byte Signed Value	8.001 DPT_Value_2_Count	unlocked	read/write	13DFFEA5-2334-4EBD-B828-C7C972BEA1A3
00/0/005	Group 00/0/005	Boolean	1.001 DPT_Switch	unlocked	read/write	A797C586-76A1-4A8D-90EB-2BFE8ACD4DA6

Group 1.1.2	Address range: [6 - 10]	GUID: BC1B7370-ABAD-4D23-A9F4-D80DD4987C78

Address	Name	Data point type	Unit		Rights	GUID
00/0/006	Group 00/0/006	Boolean	1.001 DPT_Switch	unlocked	read/write	7E14C947-82C8-4D45-B43C-D39FD02381A0
00/0/007	Group 00/0/007	Time	10.001 DPT_TimeOfDay	unlocked	read/write	EDE80D10-7410-4A82-A135-790E7FEBD4D7
00/0/008	Group 00/0/008	Date	11.001 DPT_Date	unlocked	read/write	3C1BA52E-73C7-46EE-81A0-6E6DB5B690C0
00/0/009	Group 00/0/009	Boolean	1.001 DPT_Switch	unlocked	read/write	A278A266-5016-42DE-A846-

5.2 Print Preview



The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these , markers the user has visual control over the margin settings.

JUNG Visu Pro
Calendar Program

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation systems....3
1.3 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....6
3.2 Toolbar....9
3.3 Tool Windows....9
3.4 Window Management....12

4 FUNCTIONS....15

4.1 Calendar Project....15
4.1.1 Create Calendar Project....15
4.1.2 Open Calendar Project....17
4.1.3 Delete Calendar Project....18
4.1.4 Edit Project Properties....18
4.1.5 Rename Calendar Project....19
4.2 Calendars....20
4.2.1 Create Calendar....20
4.2.2 Activate/Deactivate Calendar....21
4.2.3 Open Calendar....22
4.2.4 Delete Calendar....23
4.2.5 Edit Calendar Name....23
4.3 Day Programs....24
4.3.1 Create Day Program....24
4.3.2 Open Day Program....25
4.3.3 Delete Day Program....25
4.3.4 Edit Day Program Properties....26
4.3.5 Insert Command into Day Program....26
4.3.6 Edit Day Program....29
4.4 Calendar Configuration....35
4.4.1 Holiday Programming....36
4.4.2 Insert Day Programs Manually....38
4.5 Holiday Definitions....40
4.5.1 Define Annual Holidays as Days of the Year....40
4.5.2 Define Holiday Relative to Easter....42
4.5.3 Define Holiday as Weekday in a Month....42
4.5.4 Define Holiday Relative to Fixed Day of the Year....43
4.5.5 Define Holidays with Fixed Dates....44
4.6 The Process Model....45
4.7 Simulation....46
4.8 Applying the Calendar Project....47

5 MESSAGE LIST....47

6 REPORTS.....48
6.1 Create Report.....48
6.2 Print Preview.....51

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.
Interfaces	Interface for the connection to the process

1.2 Operation systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

After the installation of the visualization system a program group is created in the Start Menu. You can adopt the default settings or change them manually during installation.

You can start the control panel of the visualization system from the WINDOWS program manager. In order to clearly distinguish between the different applications, different colors were used. The icon of the **calendar** program contains a yellow colored bar .

To start an application, select the list entry ***Function Modules*** in the sidebar of the control panel and double-click the icon of the application you wish to start.

The **calendar** application is used to plan actions for technical processes based on day programs and calendars. It is a perpetual calendar which allows the definition of recurrent events or intervals of time for an indefinite period in the future.

First steps:

1. Create a calendar project.
2. Connect the calendar project to a process model.
3. Create a calendar and activate it.
4. Create day programs for the calendar and assign process variables to the day programs, thus defining which commands will be executed on which date and at which time for a given calendar.
5. Program the calendar: Set the standard day for the calendar, assign day programs and define holidays.

For a more detailed description of the individual steps, see section ***4 Functions***.

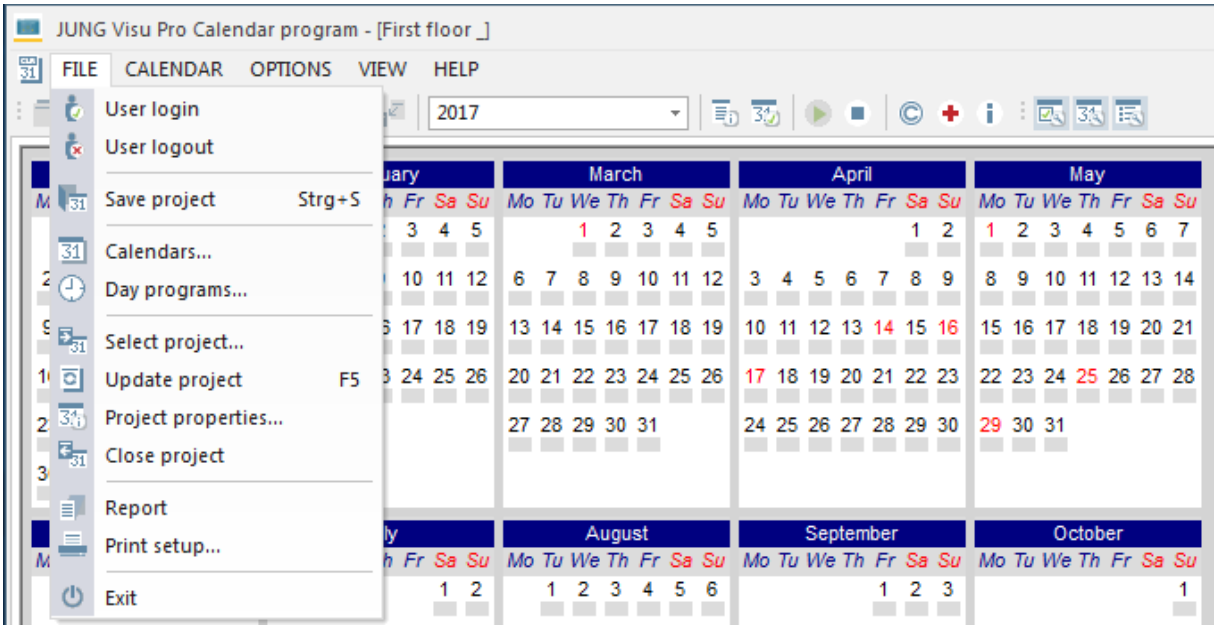
3 User Interface

The user interface of the calendar program is composed of the following main components:

- **The menu**: The options contained in the menu depend on the current configuration element loaded in the workspace (i.e. day program, calendar, holiday definitions or holiday programming).
- **The toolbar**: The toolbar contains buttons which provide quick access to certain functions.
- **Workspace**: The workspace is the main window of the calendar program where the configuration elements are loaded and edited.
- **Tool windows**: The tool windows contain functions for the creation and configuration of the time switch functions.
- **Message window**: The message window contains a list of process and system messages.

3.1 Menu

Menu item *File*



User login: Opens a dialog to log a user into the system.

User logout: Logs current user out of the system.

Save project: Saves the current project.

Calendars: Opens the dialog ***Calendar organizer*** to create new calendars and maintain existing calendars.

Day programs: Opens the dialog ***Day program organizer*** to create new day programs and maintain existing ones.

Select project: Opens a dialog to connect the current project to a logical process model.

Update project: Updates data imported from the logical process model.

Project properties: Opens a dialog to change general properties of the project.

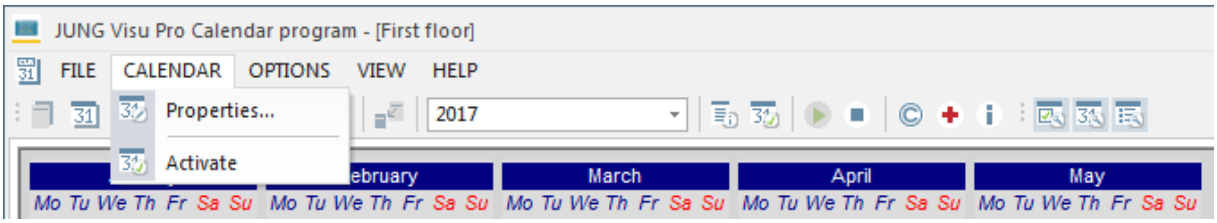
Close project: Closes the current calendar project.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Closes the editor and saves the current calendar. Shortcut Alt + F4.

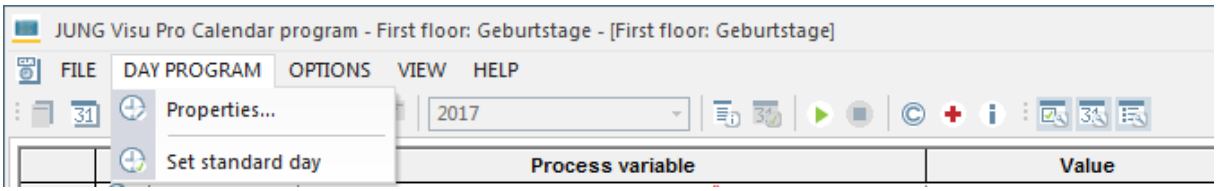
Menu item *Calendar*



Properties: Opens a dialog to change the name of the calendar.

Activate: Activates the calendar.

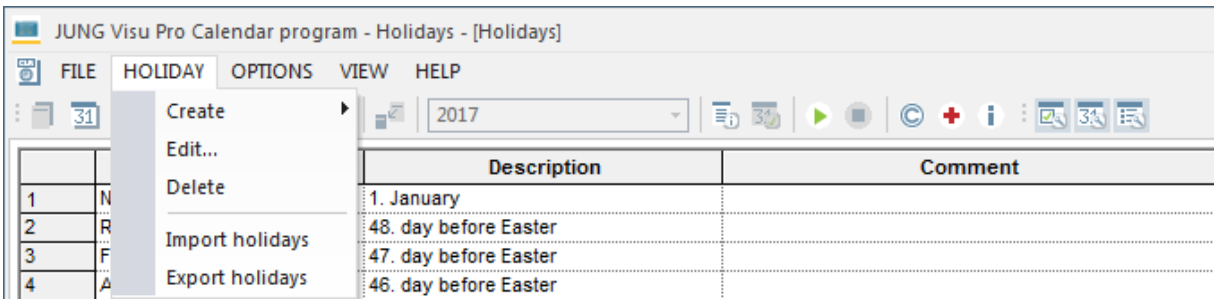
Menu item *Day program*



Properties: Opens a dialog where you can edit the name and color of the day program.

Set standard day: Defines the current day program as standard day.

Menu item *Holiday*



Create: Opens a dialog to create a new holiday or a special day.

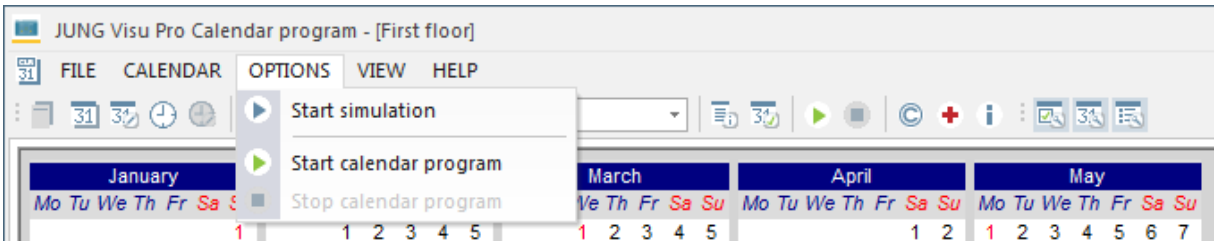
Edit: Opens a dialog to edit the selected holiday.

Delete: Deletes the selected holiday.

Import holidays: Opens a dialog to import holiday definitions from an existing file (*.esfclk).

Export holidays: Opens a dialog to save holiday definitions to a file (*.esfclk).

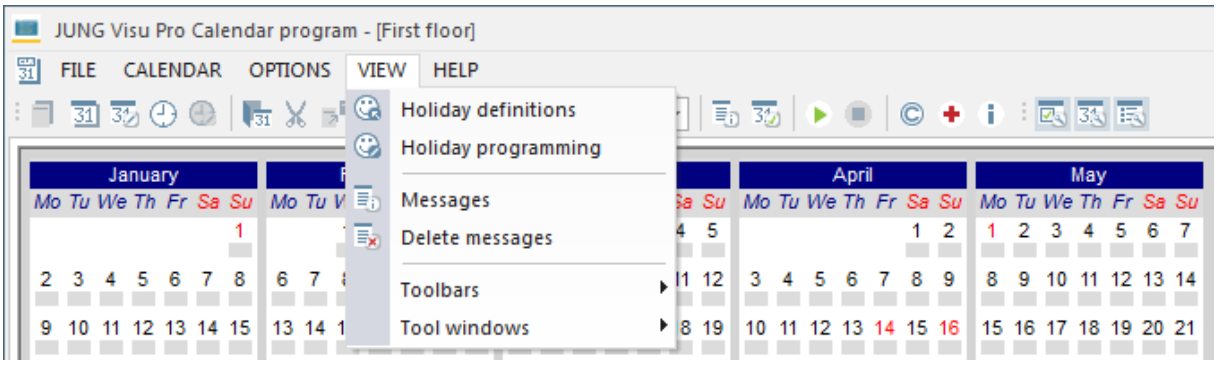
Menu item *Options*



Start simulation: Starts simulation of the calendar project.

Start calendar program: Starts the calendar project.
Stop calendar program: Stops the simulation or the running calendar project.

Menu item *View*



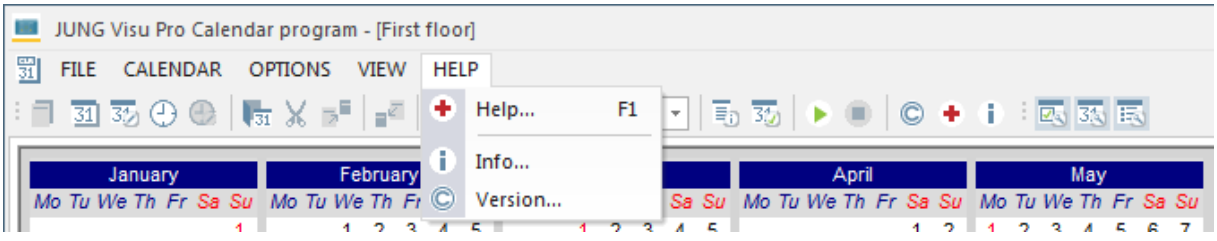
Holiday definitions: Opens a dialog to define holidays.

Holiday programming: Opens a dialog to program holidays and to assign day programs to holidays and other special days.

Messages: Toggles the visibility of the message list.

Delete messages: Deletes the contents of the message list.

Menu item *Help*



Help: Opens the help file.
Info: Shows product and license information.
Version: Shows version and copyright information.

3.2 Toolbar

Project organizer: Opens the dialog ***Project organizer***.

Calendars: Opens the dialog ***Calendar organizer***.

Edit calendar properties: Opens a dialog to edit general properties of the calendar.

Organize: Opens the dialog ***Day program organizer***.

Day program properties: Opens a dialog to edit general properties of the day program.

Save: Saves the current project.

Cut: Deletes selected items and copies them to the clipboard.

Copy: Copies selected items to the clipboard.

Paste: Pastes selected items from the clipboard.

Messages: Toggles the visibility of the message list.

Version: Shows version and copyright information.

Help: Opens the help file.

Info: Shows the product information.

Activate calendar: Toggles the activation state of the current calendar.

Start: Connects the calendar program to the KNX bus.

Stop: Interrupts the connection to the KNX bus.

Tasks tool window: Toggles the visibility of the tasks tool window.

Calendar tool window: Toggles the visibility of the calendar tool window.

Catalog tool window: Toggles the visibility of the catalog tool window.

3.3 Tool Windows

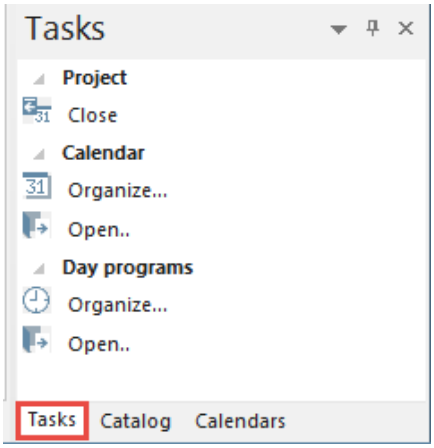
The calendar program contains different tool windows which provide tools and information according to the current state in the editor.

To toggle the visibility of the different tool windows, click the menu item ***View*** and select under ***Tool windows*** the tool window you want to show or hide.

Alternatively, use the tool windows toolbar to toggle the visibility of the tool windows. To show or hide the tool windows toolbar, click the menu item ***View*** and select ***Toolbars*** and then ***Tool windows***.

Tasks tool window

The tasks tool window provides quick access to frequently used tasks. To start a task, click on an item in the tasks list. The content of the tasks tool window depends on the configuration elements loaded in the workspace.



If no calendar project has been loaded yet:
Click ***Organize*** to start the dialog ***Project organizer*** where you can select a calendar project.

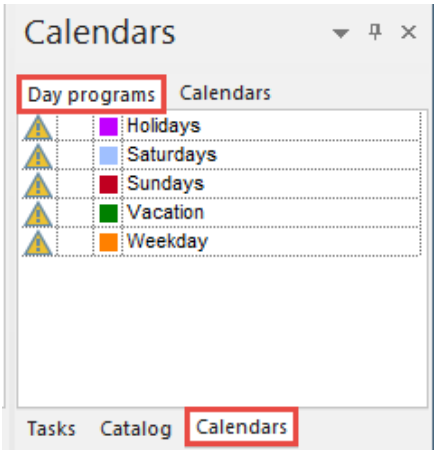
If a calendar project has been loaded:
Click ***Organize*** under the category ***Calendar*** to start the dialog ***Calendar organizer*** where you can select a calendar.

If a calendar has been loaded:
Click ***Organize*** under the category ***Day programs*** to start the dialog ***Day program organizer*** where you can select a day program.

Calendars tool window

The calendars tool window contains two tabs: The tab ***Day programs*** and the tab ***Calendars.***

The tab *Day programs:*



This tab shows a list of all day programs of the active calendar.

- The symbol ✓ in the second column indicates whether the day program has been defined as standard day. To assign the standard day to a day program, double-click in the second column in the row of the day program.

  Alternatively, use the menu item ***Day program*** when the day program is loaded in the workspace.

- To edit the color settings of the day program, double-click the color box. The dialog ***Colors*** will be opened where you can choose a color for the day program.
- To open a day program, double-click its name.
- You can insert a day program into the calendar or the holiday programming window by using the drag-and-drop function.

Problem indication

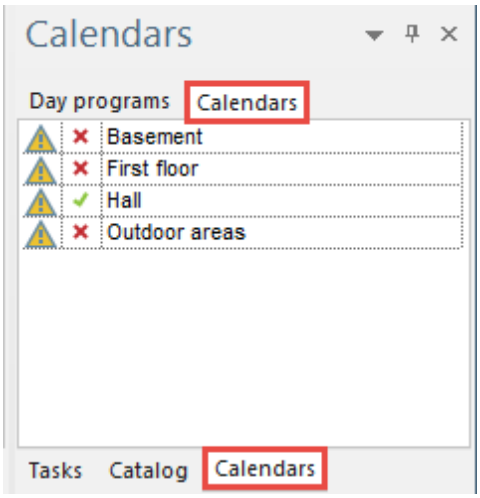
The first column is reserved to indicate possible problems.

Possible causes:

- Commands have been used for a process variable which is no longer available or the command is no longer allowed.
- The day program does not contain any commands yet.
- Two commands are sent to a single process variable at the same time. (In the day program, erroneous commands are marked with red color!)

The tab *Calendars*:

This tab shows a list of all calendars created in the calendar project. It has the following properties:



- The second column shows the activation state of the calendar. The symbol ✓ represents the active state of the calendar. The symbol ✗ indicates that the calendar is not active.

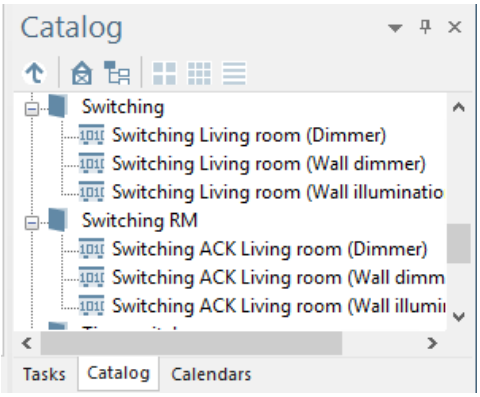
  To change the activation state of a calendar, double-click this column. Alternatively, use the menu item ***Calendars*** or the symbol in the toolbar.

- To open the calendar, double-click the calendar name.

Problem indication

The first column is reserved to indicate possible problems.

Possible causes: The calendar comprises day programs which contain erroneous or incomplete commands. (In the day program erroneous commands are marked with red color!)



Catalog tool window

The catalog tool window comprises the connected process models with the corresponding process variables. To insert a command into a day program, drag the process variable to the day program window.

3.4 Window Management

You can undock tool windows and the message window from their positions and

- arrange them anywhere in the calendar program editor as **floating windows** or
- **dock** them to predefined positions within the calendar program editor or within existing windows.

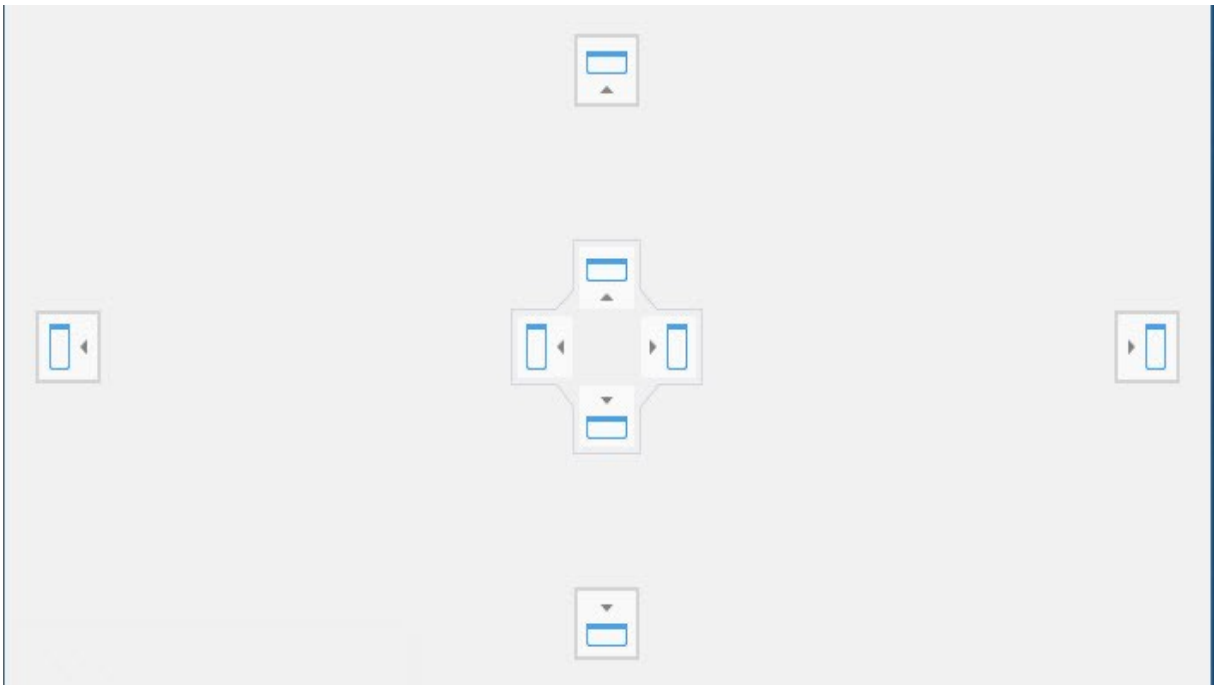
Floating windows

You can drag a window to any position within or outside the calendar program editor. Click the title bar of the window you wish to rearrange and drag it to the desired position in the editor. The window exists now as a separate floating window.

To restore the window to its initial position, double-click its title bar.

Docked windows

With the help of **docking guides** you can dock windows to predefined positions in the calendar program. The docking guides are displayed when you start dragging a window from its position. They indicate where the window may be docked. The number of the possible docking positions depends on the target window and on the window you want to dock.



There are two options for docking a window:

- relatively to the entire program editor: for this purpose, docking guides are displayed at the top, bottom, left and right side of the editor edge, indicating the possible docking positions in the entire editor area;

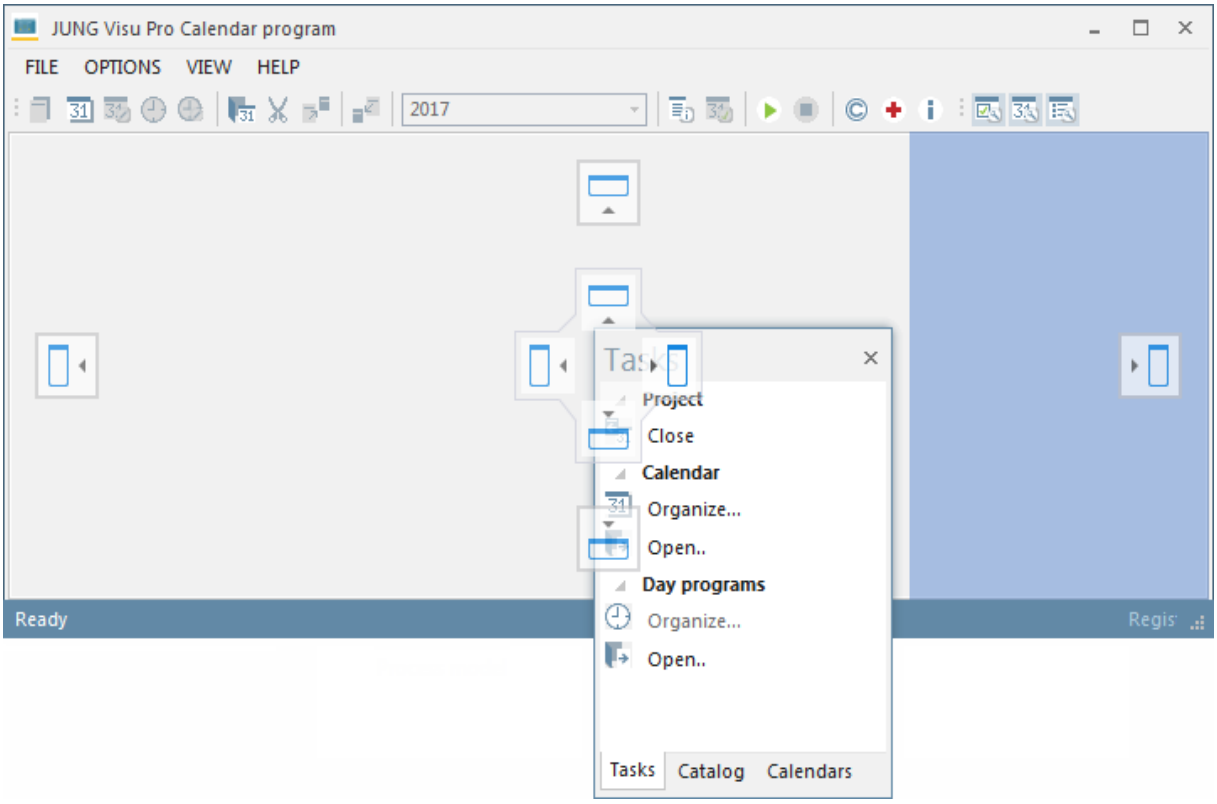
- relatively to an existing window: for this purpose, docking elements are displayed in the center of the window the mouse cursor is currently over, indicating the possible docking positions within this window.

How to dock a window:

- Click the window's title bar with the left mouse button, keep the mouse button down and start dragging the window to the area you wish to dock it to.

- When you start dragging the window, docking guides are displayed indicating the possible docking positions within the target window.

- When you move the mouse cursor over one of the docking guides a semi-transparent blue rectangle is displayed indicating the dropping area. This is the area the window will be docked to if you release the mouse button.

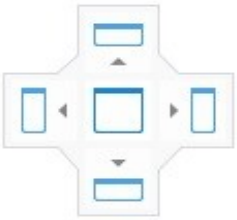


- To position the window in the highlighted area, release the mouse button.

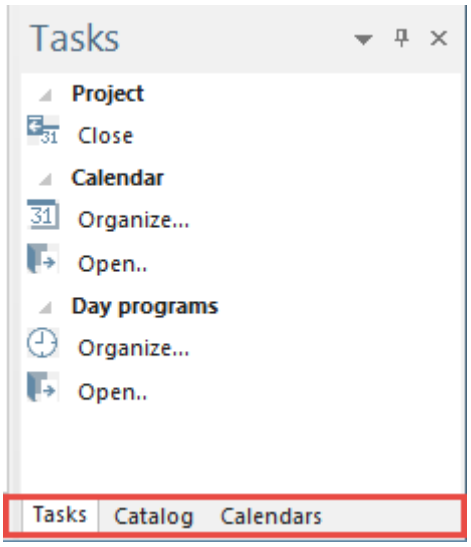
Tabbed windows

Tool windows can be docked to each other to form tabbed groups. To insert a window as a tab page within an existing window:

- Click the window's title bar, hold down the mouse button and drag the window to the target window.



- During the dragging operation docking guides are displayed, indicating the possible docking positions within the target window. The center docking guide represents a tabbed group.
- Release the mouse button over the central docking guide to insert the window as a new tab page into the existing window.

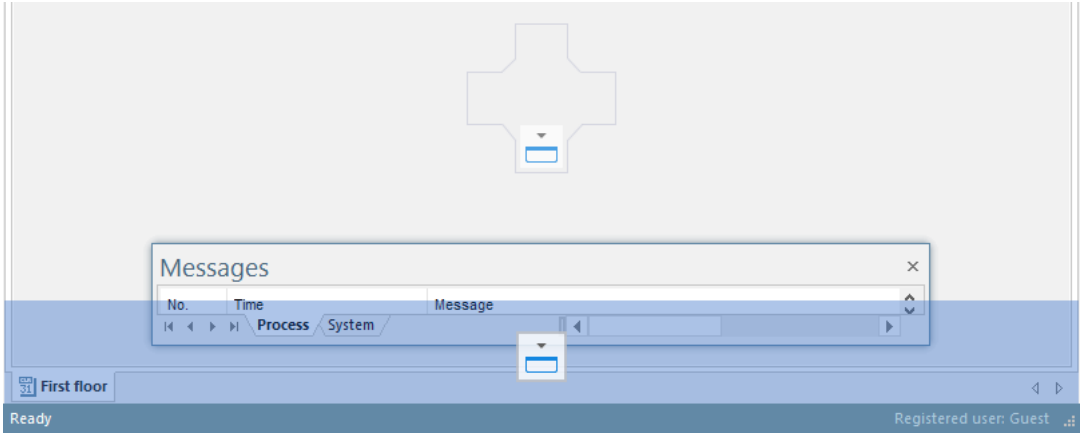


The tool windows docked in the tabbed group are displayed as tabs in the tab bar at the bottom of the window.

To remove a window from its arrangement as tabbed window, click its tab and drag it out of the tabbed group.

Docking the message list:

The message list may be docked to the bottom of the workspace edge or to the bottom of the editor edge.



4 Functions

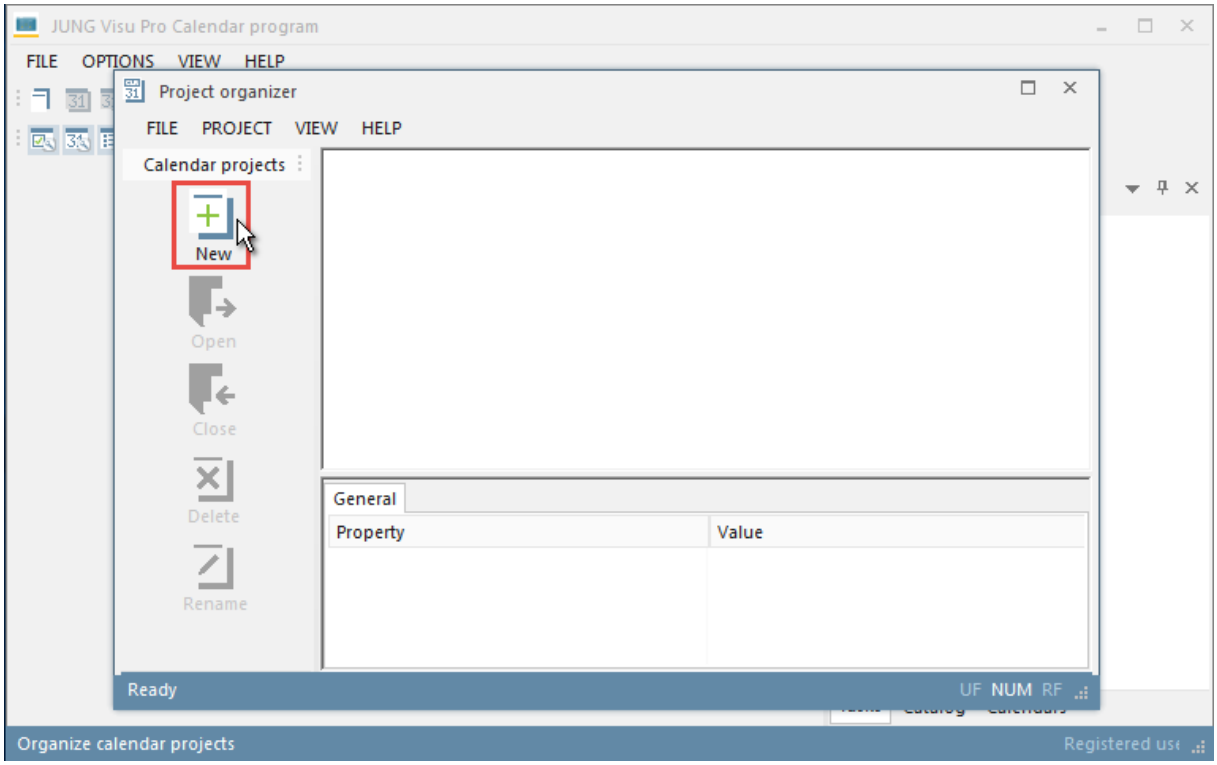
4.1 Calendar Project

4.1.1 Create Calendar Project

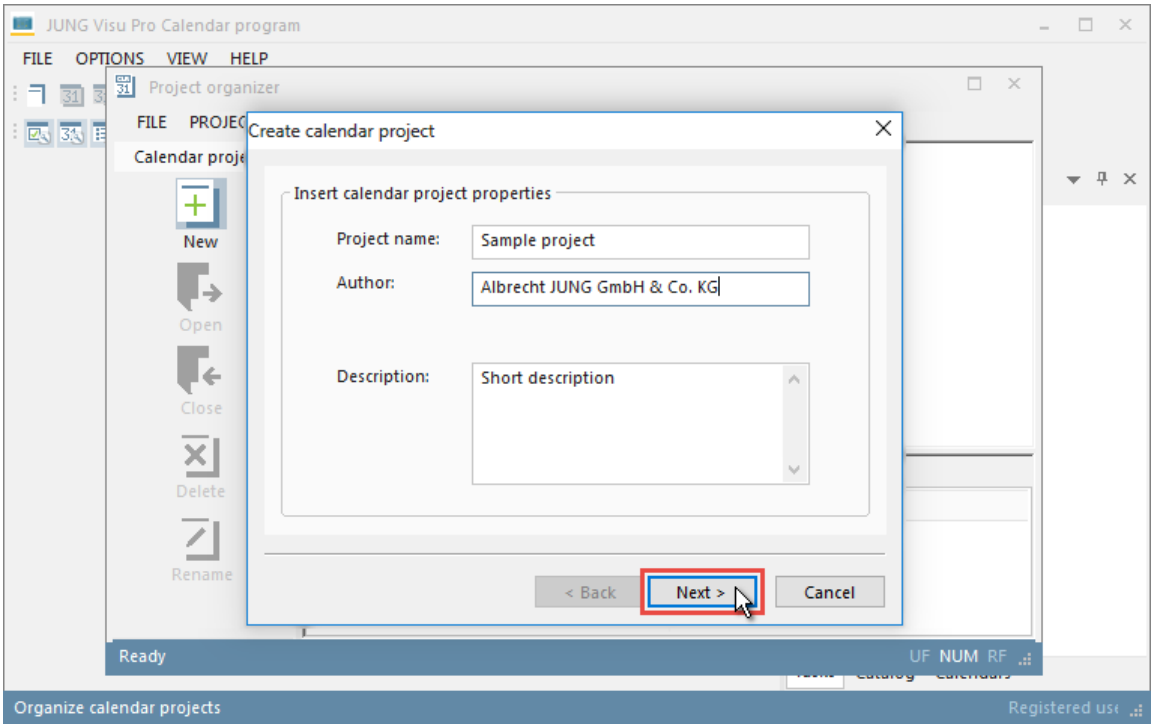
After having started the calendar program, open the dialog ***Project organizer***. For this purpose,

- choose the menu item ***File*** and click ***Projects*** or
- click ***Organize*** under the entry ***Projects*** in the tasks tool window.

In the project organizer dialog you can create, open, delete or rename calendar projects.

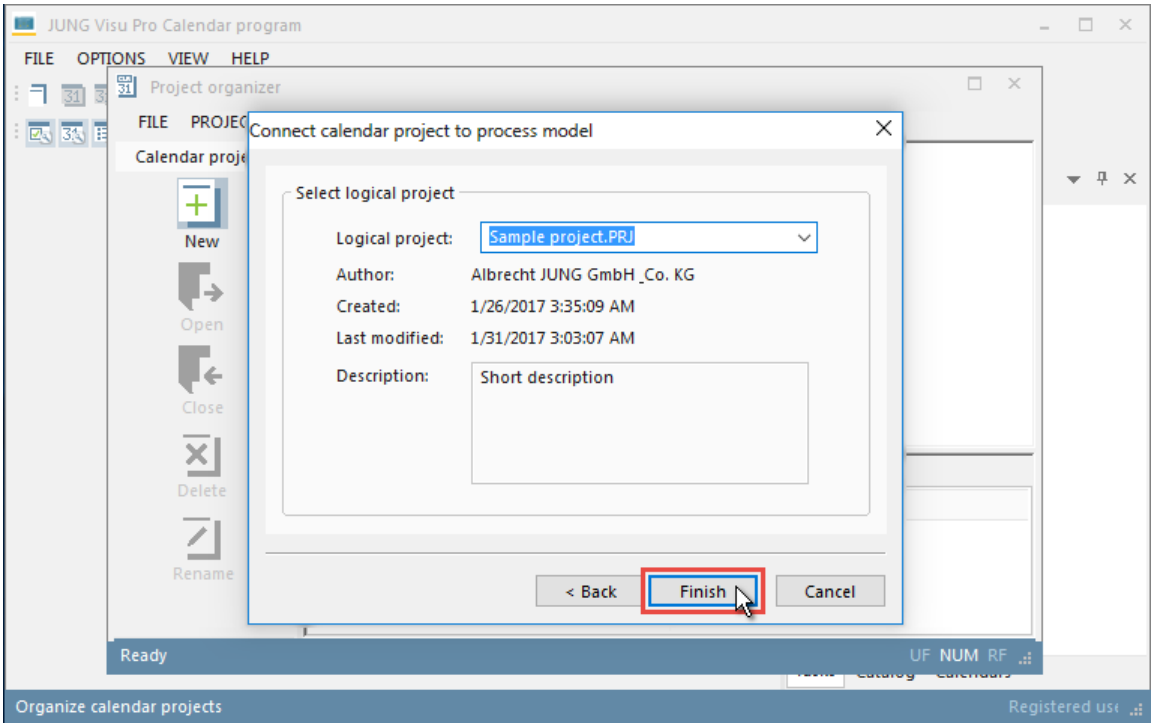


Click the button ***New*** to open the dialog ***Create calendar project***. Enter the data of the calendar project:



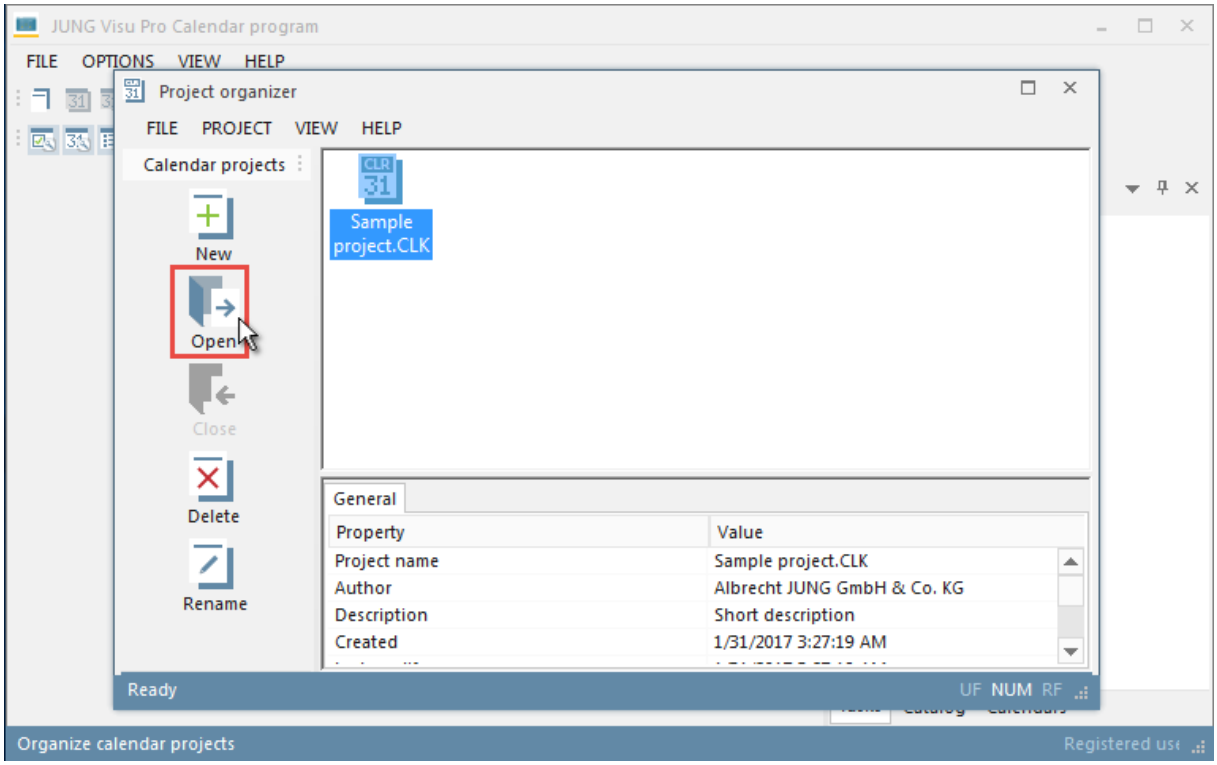
Project name: Unique name of the calendar project.
Author and description: The specifications *author* and *description* are useful but not mandatory. You can enter or change these data later.

Click ***Next*** to proceed with the creation of the calendar project. Select the process model you want to connect the calendar program to and click ***Finish*** to complete the creation of the calendar project.



The calendar project is now displayed in the project organizer and can be loaded into the calendar program.

4.1.2 Open Calendar Project



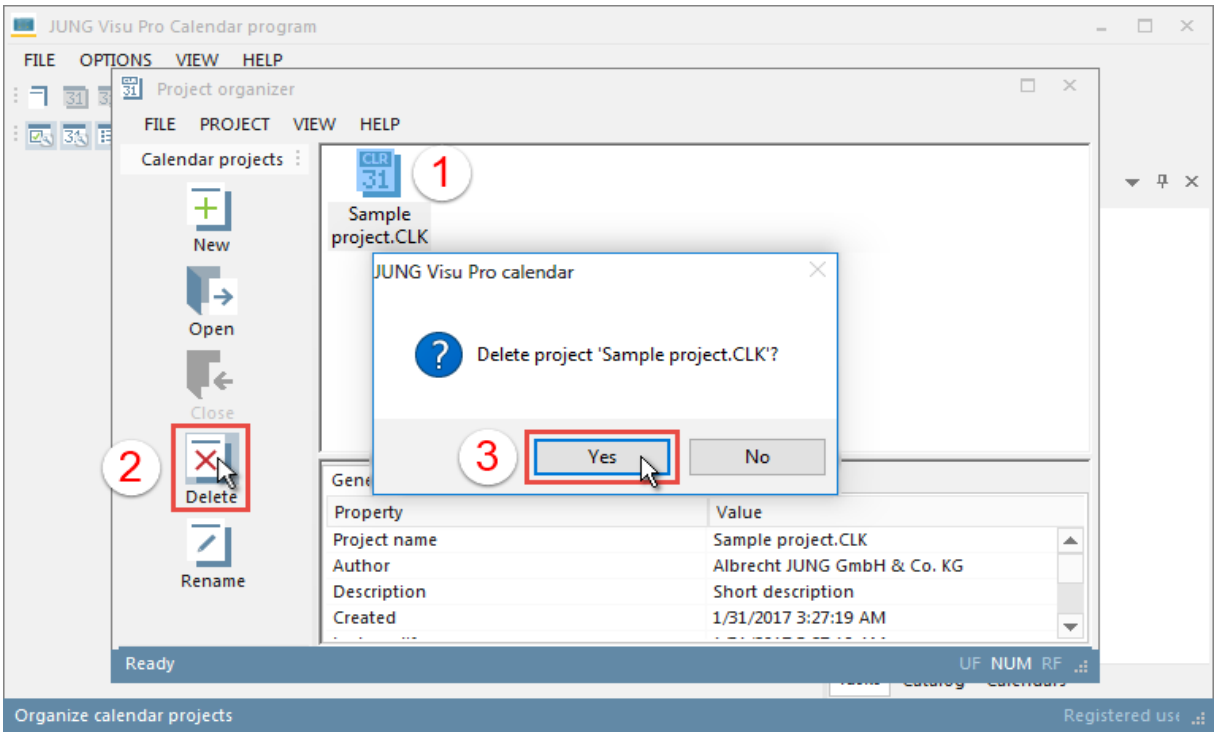
To open a project:

- double-click the project symbol or
- select the project and click the ***Open*** symbol in the sidebar or
- select the project and click the menu item ***Project > Open***.

4.1.3 Delete Calendar Project

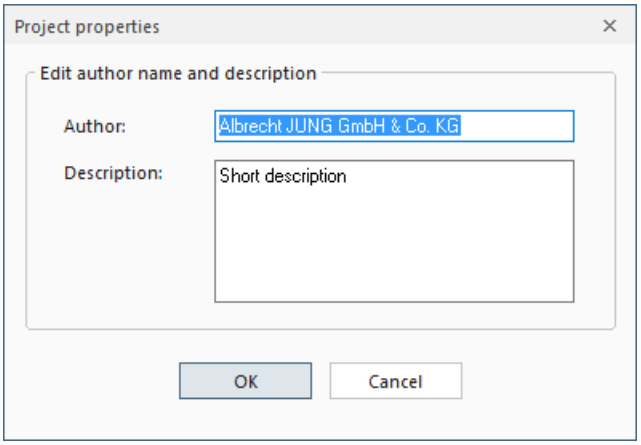
To delete a project, open the project organizer. Then

- select the project and click the ***Delete*** symbol in the sidebar of the project organizer or
- select the project and click the menu item ***Project > Delete***.



4.1.4 Edit Project Properties

Choose the menu item ***File*** and click ***Project properties***. The dialog ***Project properties*** opens.



Enter the author and a description and click ***OK*** to commit the changes and to close the dialog.

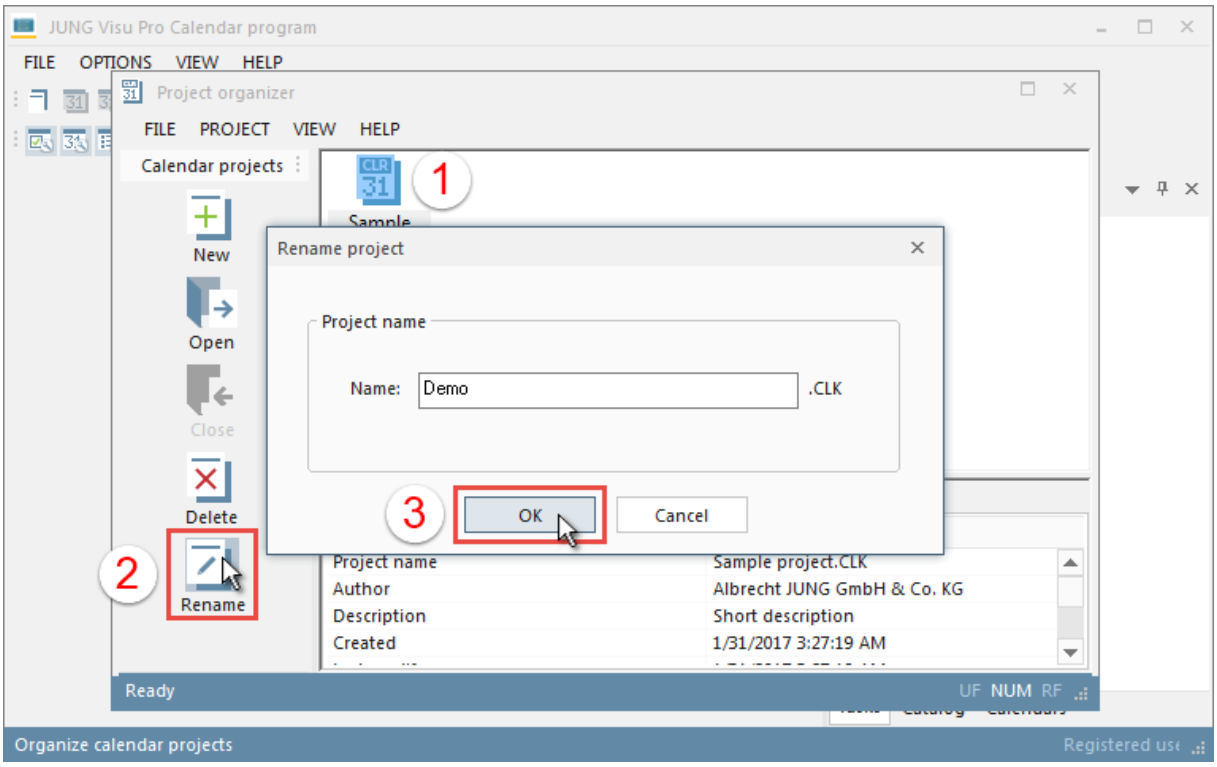
4.1.5 Rename Calendar Project

To rename a calendar project, open the project organizer. Note that the project must be closed before you can rename it.

Select the project in the project organizer and

- click the ***Rename*** symbol in the sidebar or
- click the menu item ***Project > Rename***.

The dialog ***Rename project*** opens.



Enter the new name of the project and click ***OK*** to commit the changes and to close the dialog.

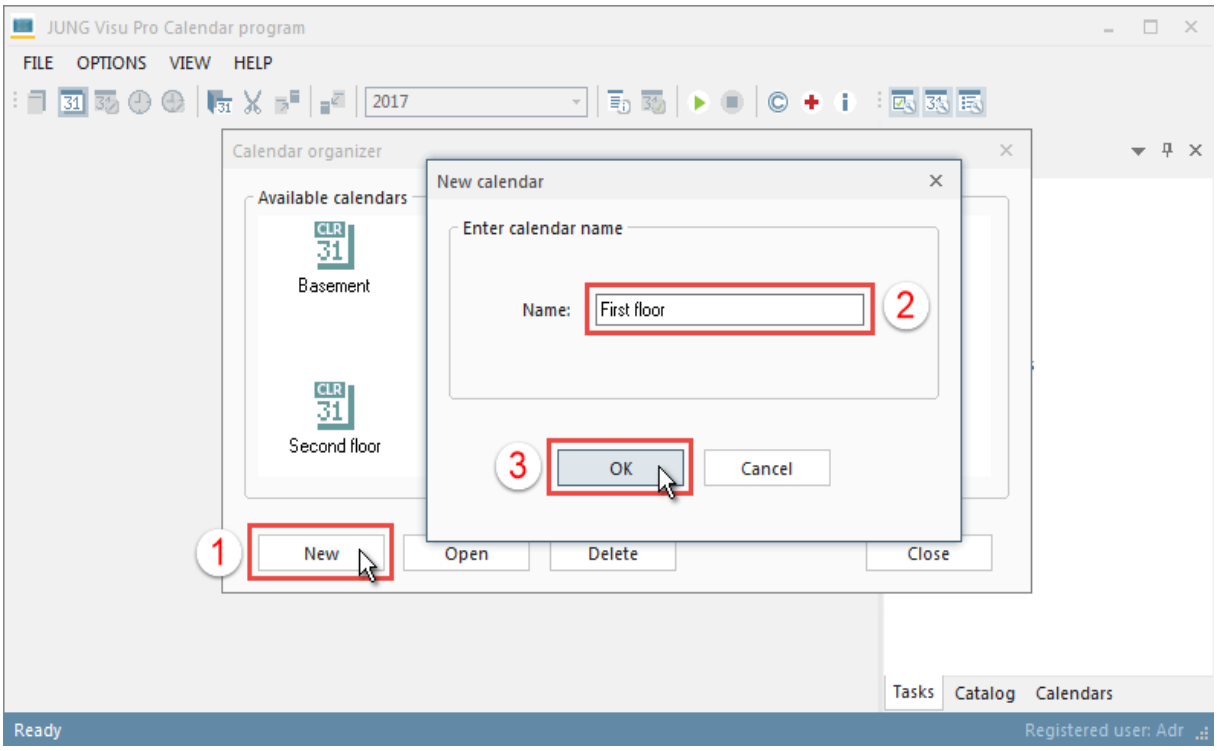
4.2 Calendars

4.2.1 Create Calendar

A calendar project may contain several calendars. To create a calendar, open first the ***Calendar organizer***. For this purpose,

- choose the menu item ***File*** and click ***Calendars*** or
- click ***Organize*** under the entry ***Calendars*** in the tasks tool window or
- click the button in the toolbar.

In the ***Calendar organizer*** you can create, delete or rename calendars of the current calendar project. To create a new calendar, click the button ***New***. The dialog ***New calendar*** opens.



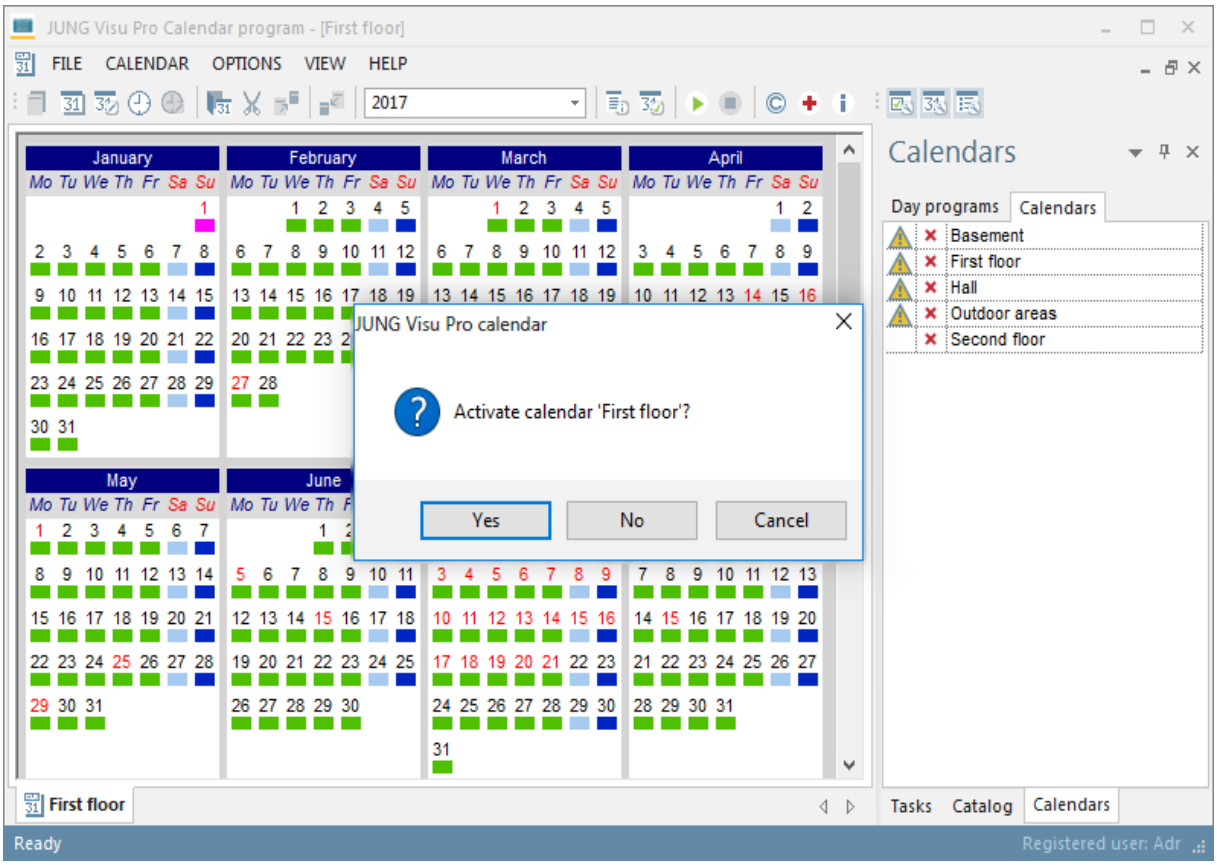
Enter a unique calendar name and click ***OK*** to close the dialog. The newly created calendar is now displayed in the dialog ***Calendar organizer*** and can be opened.

4.2.2 Activate/Deactivate Calendar

A calendar can be activated or deactivated. If a calendar is not active, its time schedule will not be evaluated.

To activate or deactivate a calendar:

- choose the menu item ***Calendar*** and click ***Activate*** or
- click the button in the toolbar or
- select in the calendar tool window the tab ***Calendars*** and double-click the second column before the calendar name. The symbol ✓ marks the activated and the symbol ✗ marks the deactivated state of the calendar. A confirmation prompt appears.



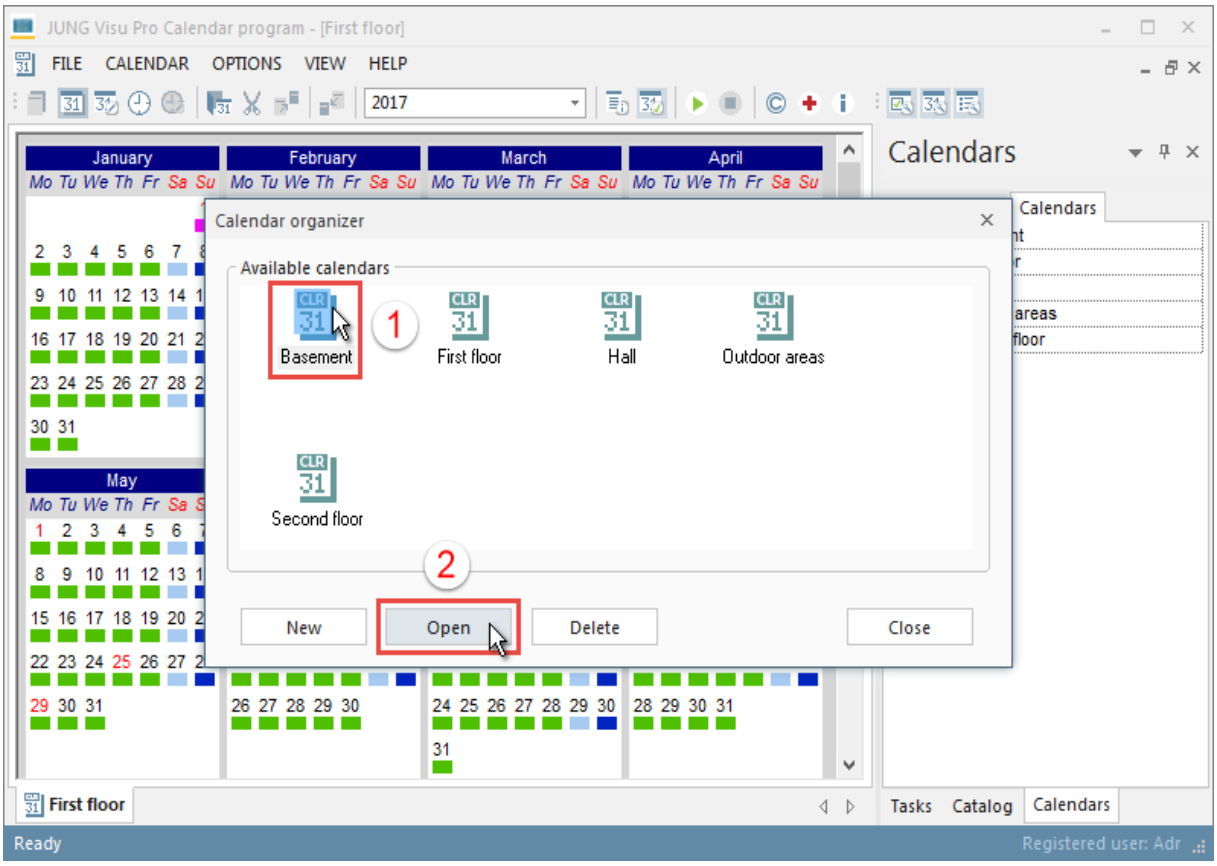
Click

- ***Yes*** to activate the calendar,
- ***No*** to deactivate the calendar and
- ***Cancel*** to cancel the operation.

The activation state is shown in the tool window ***Calendars*** on the tab ***Calendars***.

4.2.3 Open Calendar

To open a calendar, open the dialog ***Calendar organizer***.

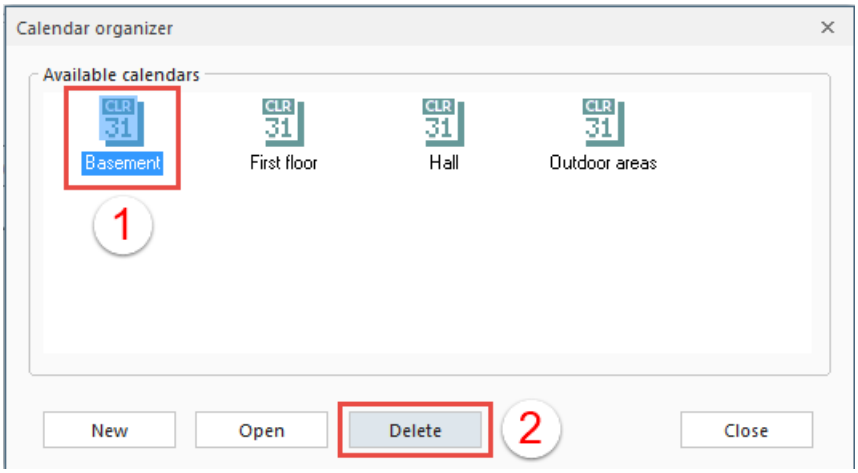


Select the calendar you want to open and click the button ***Open***. Alternatively, double-click the calendar.

You can also open a calendar by selecting the tab ***Calendars*** in the calendar tool window and double-clicking the desired calendar name.

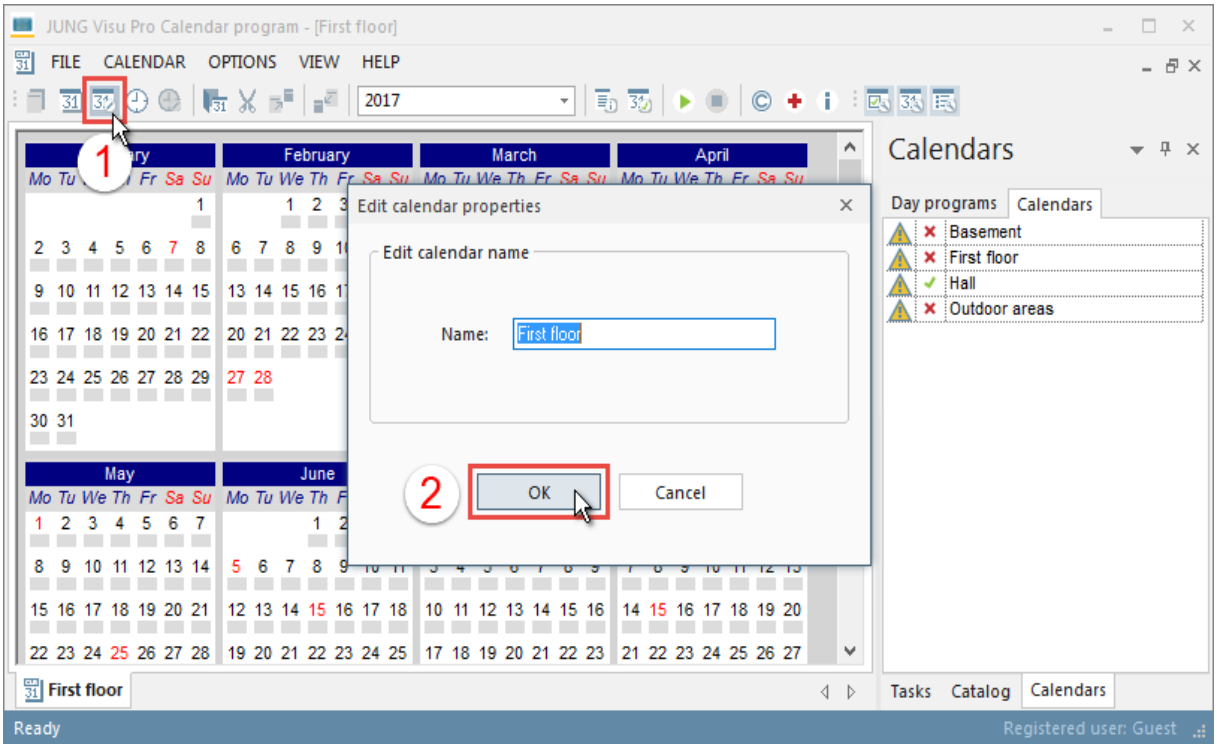
4.2.4 Delete Calendar

To delete a calendar, open the calendar organizer. Select in the calendar organizer the calendar you want to delete and click the button ***Delete***. ***Note: You will not be prompted to confirm the delete operation!***



4.2.5 Edit Calendar Name

To change the name of the calendar, choose the menu item ***Calendar*** and click ***Properties*** or click the button in the toolbar. The dialog ***Edit calendar properties*** opens.



Enter the new name of the calendar and click ***OK*** to commit the change.

4.3 Day Programs

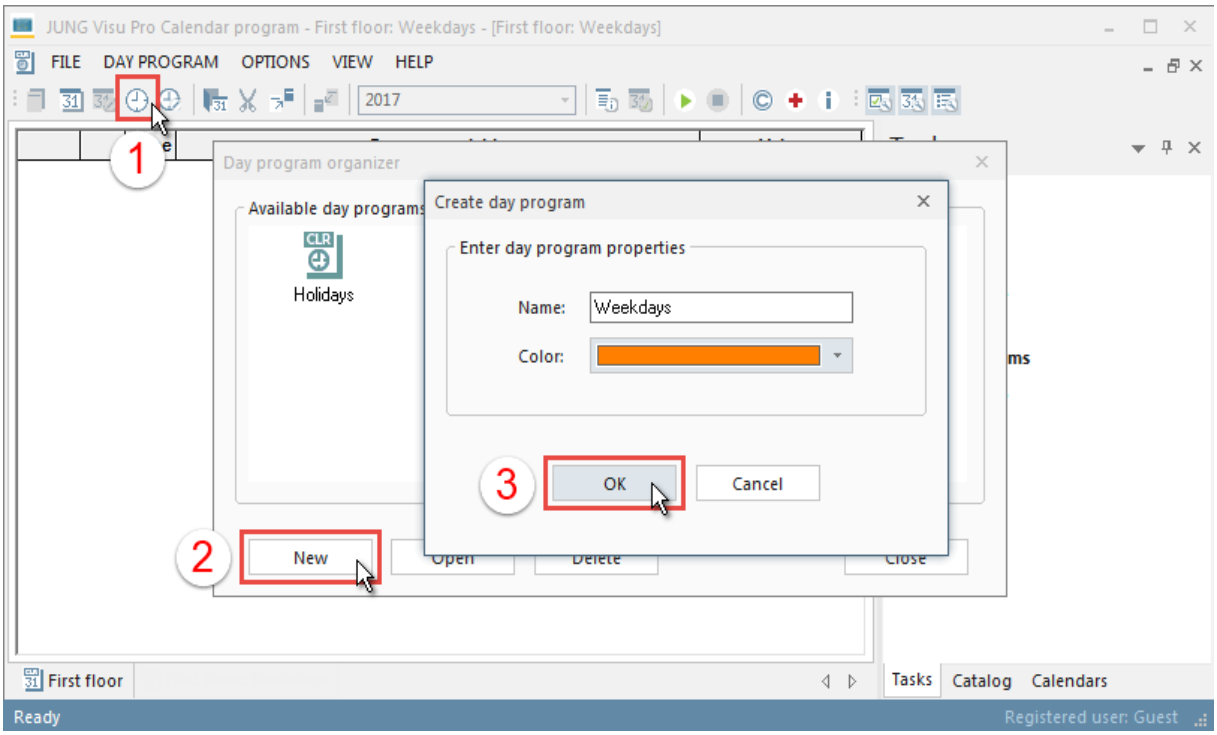
4.3.1 Create Day Program

To create a day program, open the dialog ***Day program organizer***. For this purpose,

- choose the menu item ***File*** and click ***Day programs*** or
- click ***Organize*** in the tasks tool window under the category ***Day programs*** or
- click the icon in the toolbar.

The dialog ***Day programs management*** opens where you can create new day programs and open or delete existing ones.

To create a new day program, click the button ***New***.



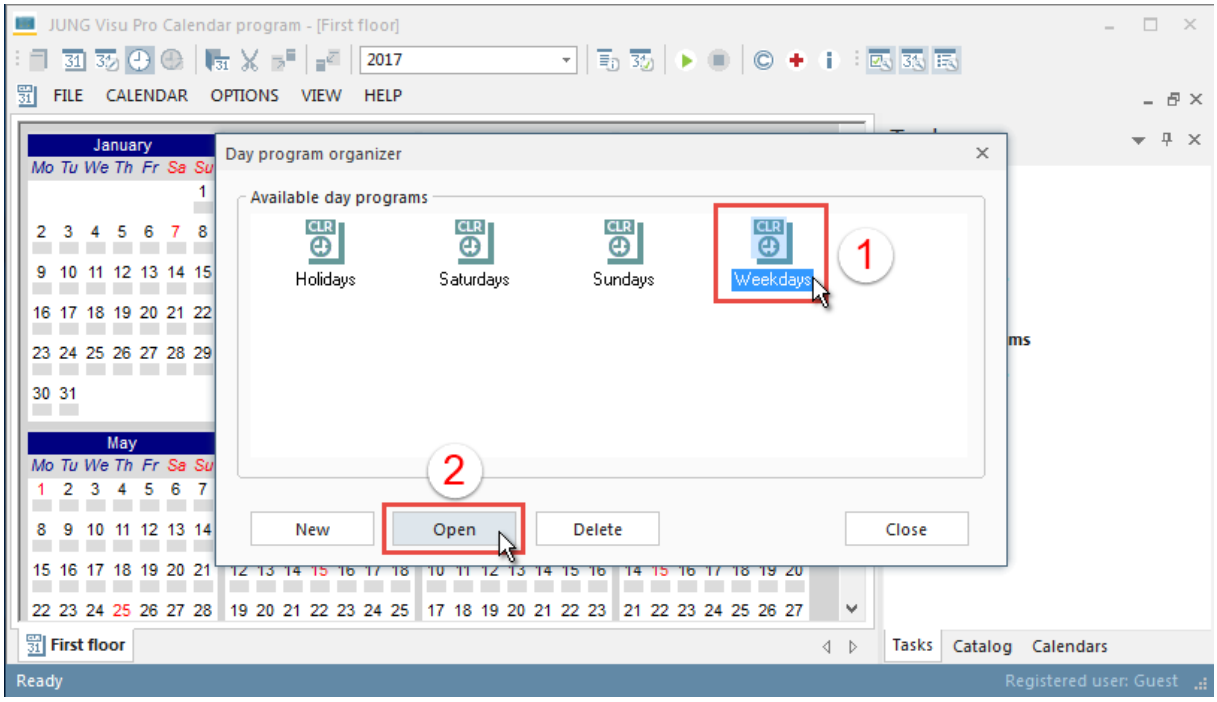
The dialog ***Create day program*** opens.

Enter the name of the day program and choose a color for it. In the calendar view the day program will be displayed in this color. Click ***OK*** to create the day program and to close the dialog.

The day program is now available in the dialog ***Day program organizer***.

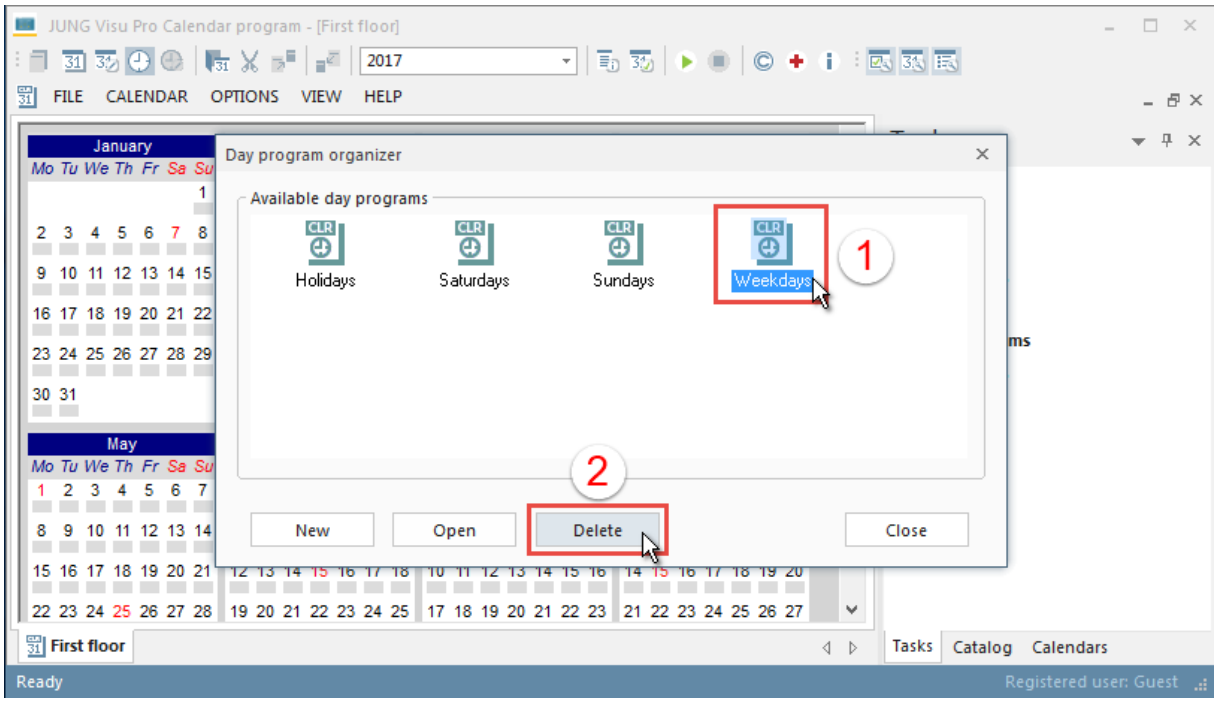
4.3.2 Open Day Program

In the ***Day program organizer*** double-click the day program symbol or select the day program and click the button ***Open***.



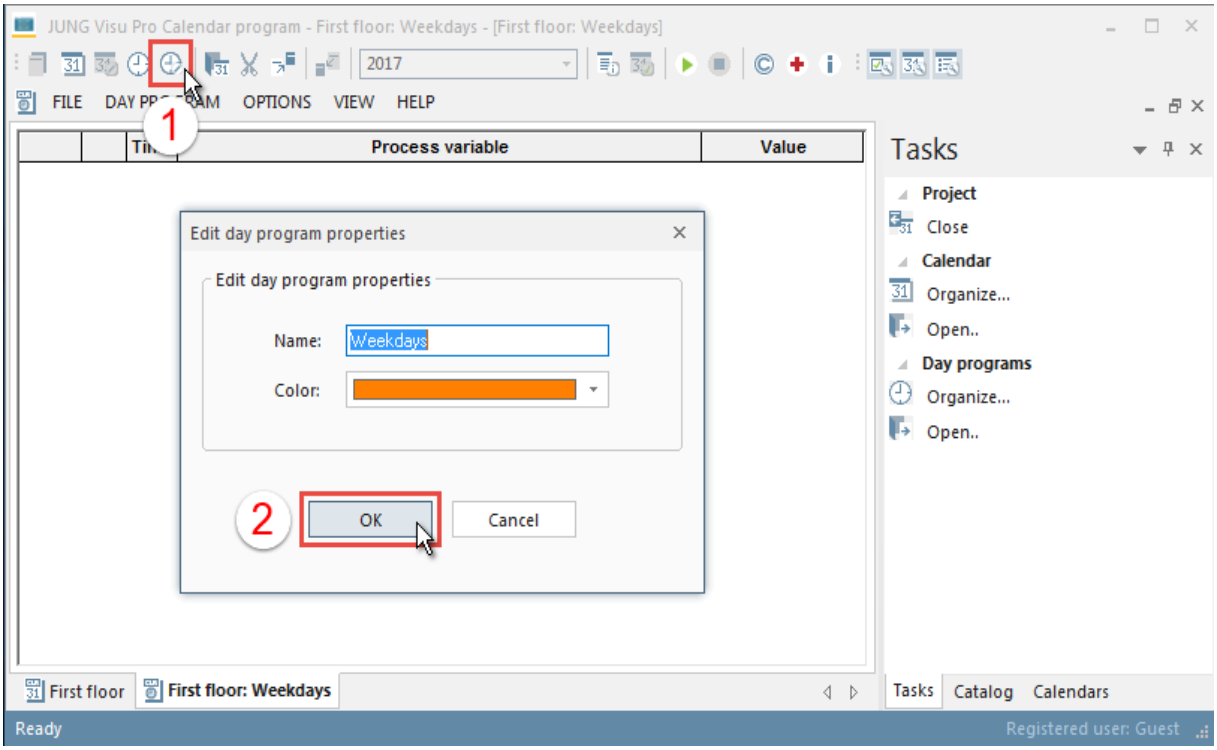
4.3.3 Delete Day Program

In the ***Day program organizer*** select the day program you want to delete and click the button ***Delete***. ***Note: You will not be prompted to confirm the delete operation!***



4.3.4 Edit Day Program Properties

Choose the menu item ***Day program*** and click ***Properties***. The dialog ***Edit day program properties*** opens.

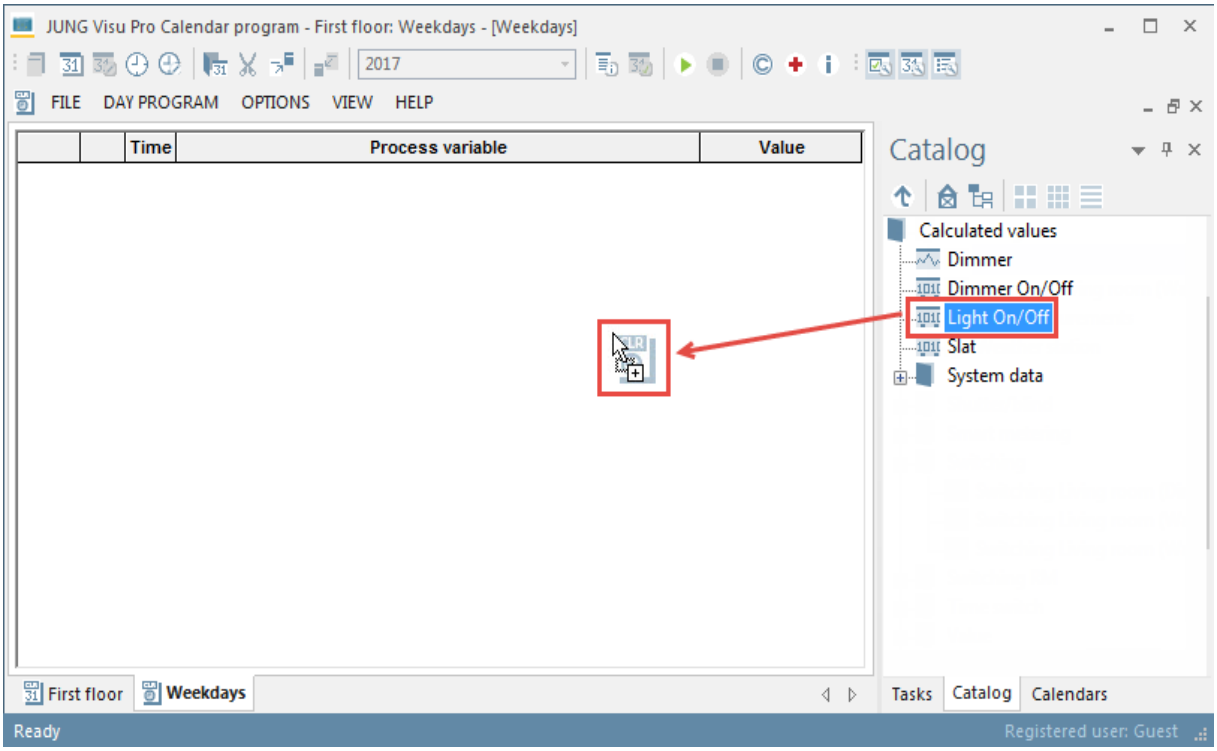


Enter a new name or choose a different color and click ***OK*** to commit the changes and to close the dialog.

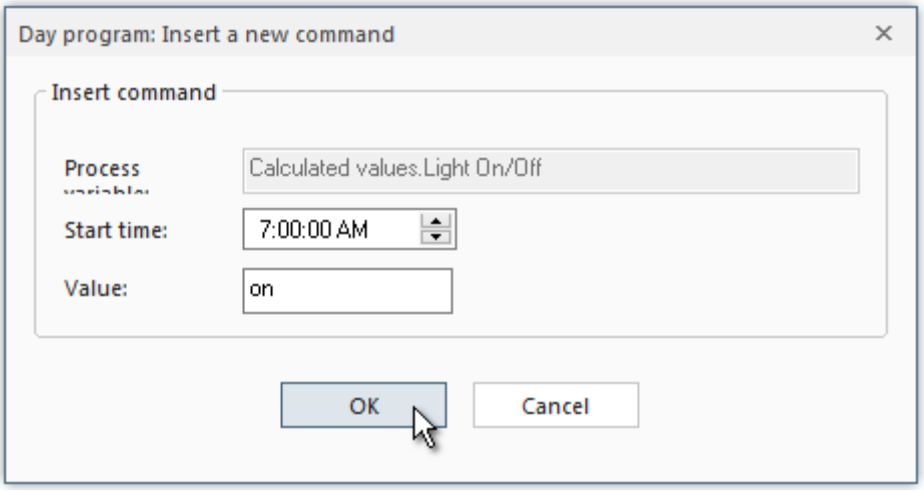
4.3.5 Insert Command into Day Program

How to insert a command into the day program:

- Load the day program into the workspace.
- In the tool window ***Catalog*** double-click the ***Process data*** symbol. The catalog will switch to a tree view containing the process variables of the connected process model.
- Use the drag and drop function to drag the desired process variable to the opened day program.



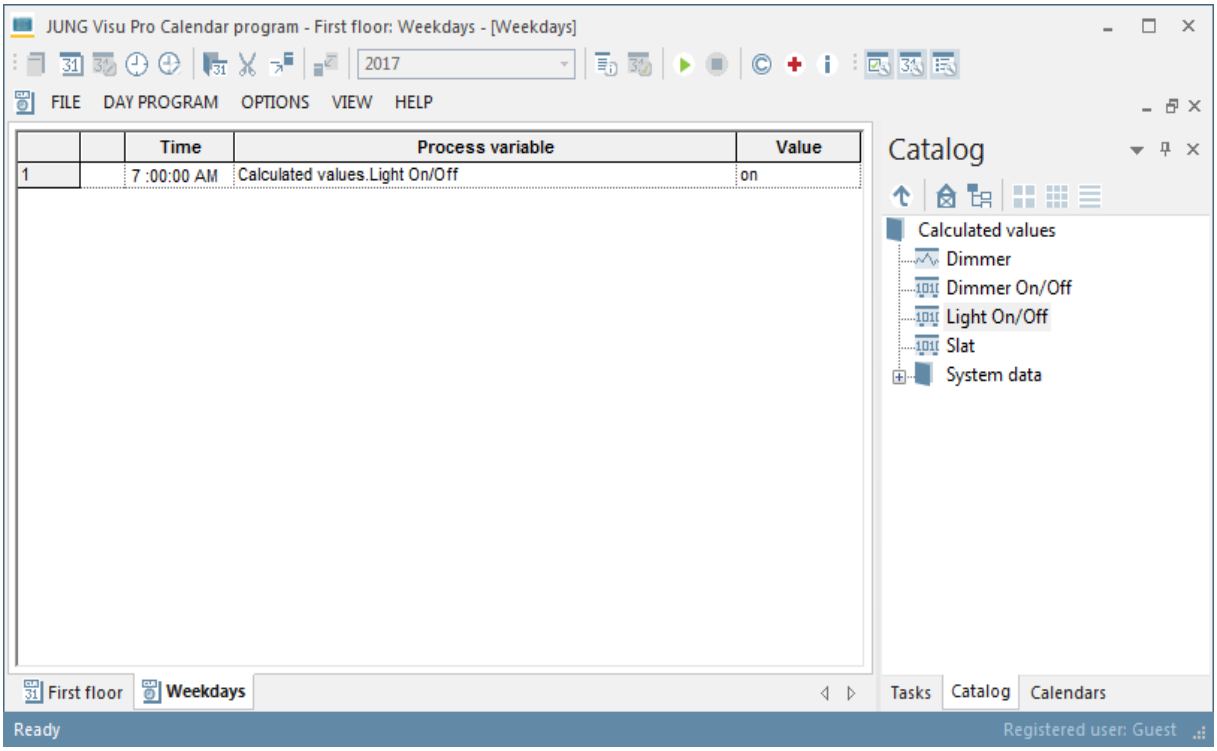
Releasing the mouse button over the day program will insert a command for the process variable in the day program command list. A dialog is opened that allows to specify the details of the inserted command.



Enter the time and value of the command. At the specified time the value will be sent to the process variable.

For binary variables the following entries are possible: **on (1)**, **off (0)** or **read value (le)**. Then press ***OK*** to commit the changes and to close the dialog. If you press ***Cancel***, no command will be inserted.

The command is displayed in the command list of the day program.



4.3.6 Edit Day Program

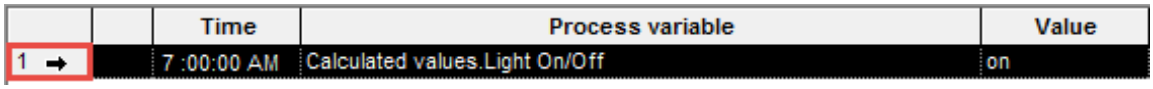
In order to edit the day program, first open the day program in the workspace.

The day program can be opened in the following ways:

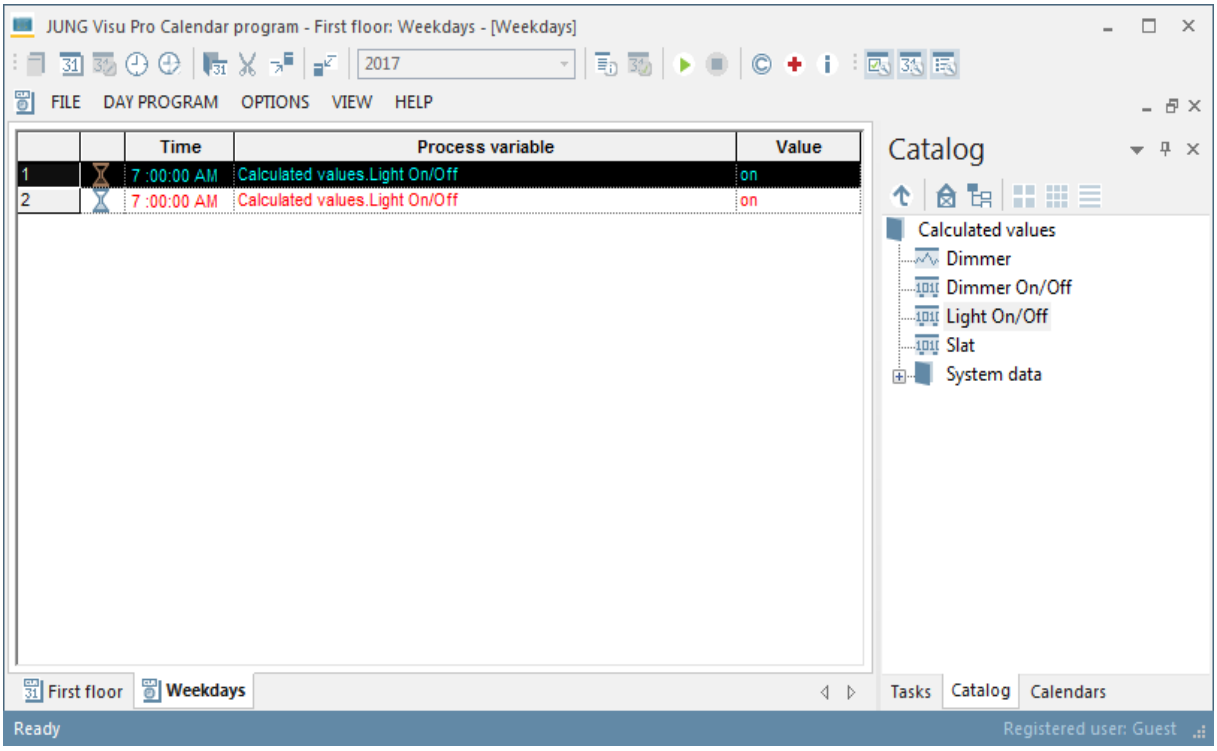
- Choose in the ***Tasks*** tool window the category ***Day programs*** and click ***Organize***. Or click the button in the toolbar. The dialog ***Day program organizer*** opens. Double-click here on the symbol of the day program that you want to edit.
- Choose in the ***Calendars*** tool window the tab ***Day programs*** and double-click the desired day program.
- Double-click the day program in the calendar.

How to copy a day command:

Click on the first column of a day command to select the process variable.



Click the button ***Copy*** in the toolbar or the keyboard keys Ctrl + C to copy the selected command to the clipboard. Then click the button ***Paste*** in the toolbar or the keyboard keys Ctrl + V to paste the day program to the day program.



Note: The symbol indicates that two commands are sent to a single variable at the same time.

		Time	Process variable	Value
1		7 :00:00 AM	Calculated values.Light On/Off	on
2		11:00:00 AM	Calculated values.Light On/Off	on

Click with the left mouse button on the time field and set the desired time.

Note: The symbol indicates that a process variable could not be found. This situation cannot occur when a new day command has been inserted, because only valid day commands can be created. However, this situation may occur when the process model has been changed. For the following example the process variable *switch* has been deleted in the logical project.

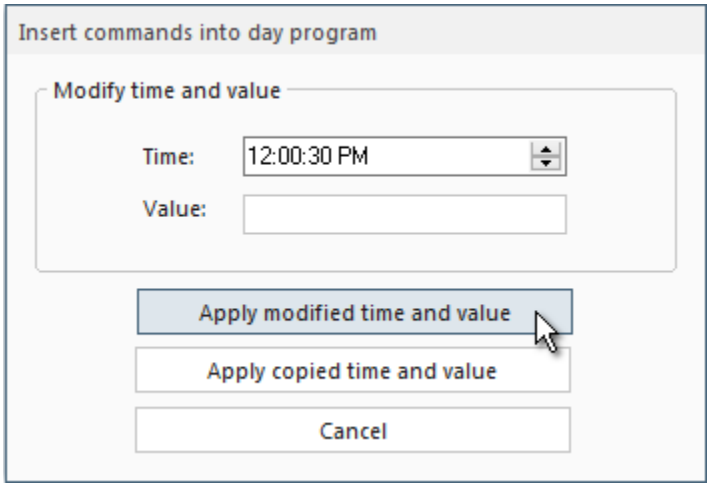
		Time	Process variable	Value
1		7 :00:00 AM	Calculated values.Light On/Off	on
2		7 :00:00 AM	Calculated values.Slat	Read value
3		3 :00:00 PM	Calculated values.Light On/Off	off

Multiple selections of day commands:

- Click on the first column of a row and move the mouse vertically while holding down the left mouse button.
- Click on the first column of a row, hold down the Shift key and click on the first column of the last row you want to select.
- Select different rows while holding down the Ctrl key.

		Time	Process variable	Value
1		7 :00:00 AM	Calculated values.Light On/Off	on
2		3 :00:00 PM	Calculated values.Light On/Off	off

The clipboard commands always apply to the whole selection. Hence you may select multiple commands and paste them to a day program. In particular you can use the clipboard to copy day commands of one day program to another day program.



If you paste day commands in the clipboard to a day program, a dialog is opened that allows you to modify the time and the value of the day commands you want to insert. After having modified the data, click the button ***Apply modified time and value***.

If you click ***Apply copied time and value***, the value or time changes made will be abandoned and the data will be inserted from the clipboard.

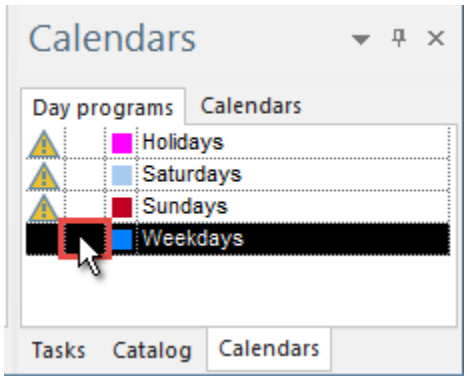
If you click ***Cancel***, the copy operation will be canceled.

For analog variables, enter a number for the value. For binary variables, use **on (1)**, **off (0)** or **read value (re)**.

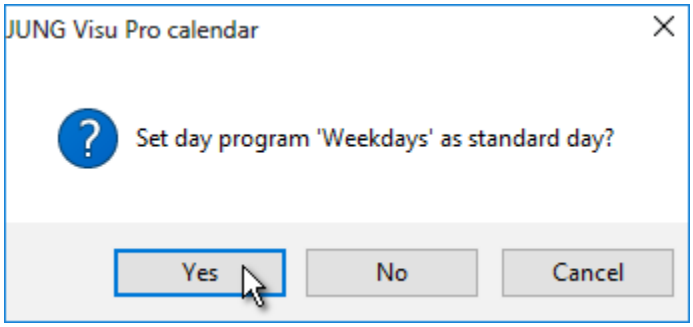
		Time	Process variable	Value
1		12:00:00 AM	Calculated values.Light On/Off	off
2		12:30:00 AM	Calculated values.Light On/Off	on
3		7 :00:00 AM	Calculated values.Light On/Off	on
4		3 :00:00 PM	Calculated values.Light On/Off	off

How to delete a day command: Select the day command and press the keyboard key *Delete*.

How to set a day program as standard day: Choose the tab ***Day programs*** in the Calendars tool window and double-click the second column in front of the day program you wish to set as standard day.



The dialog ***Calendar*** opens where you are prompted to confirm that you want to set the day program as standard day.



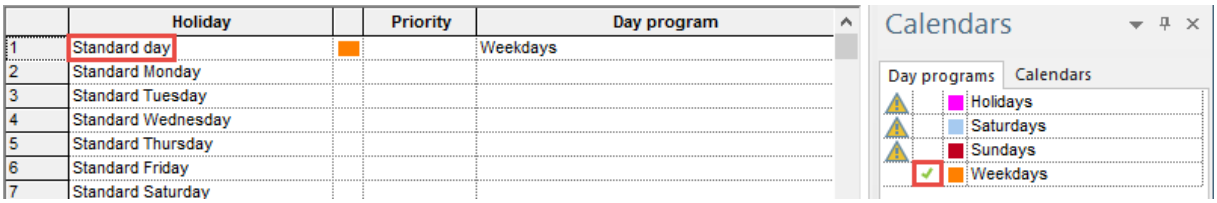
Click

- Yes to set the day program as standard day,
- No if you do not want to set the day program as standard day and
- Cancel to cancel the operation.

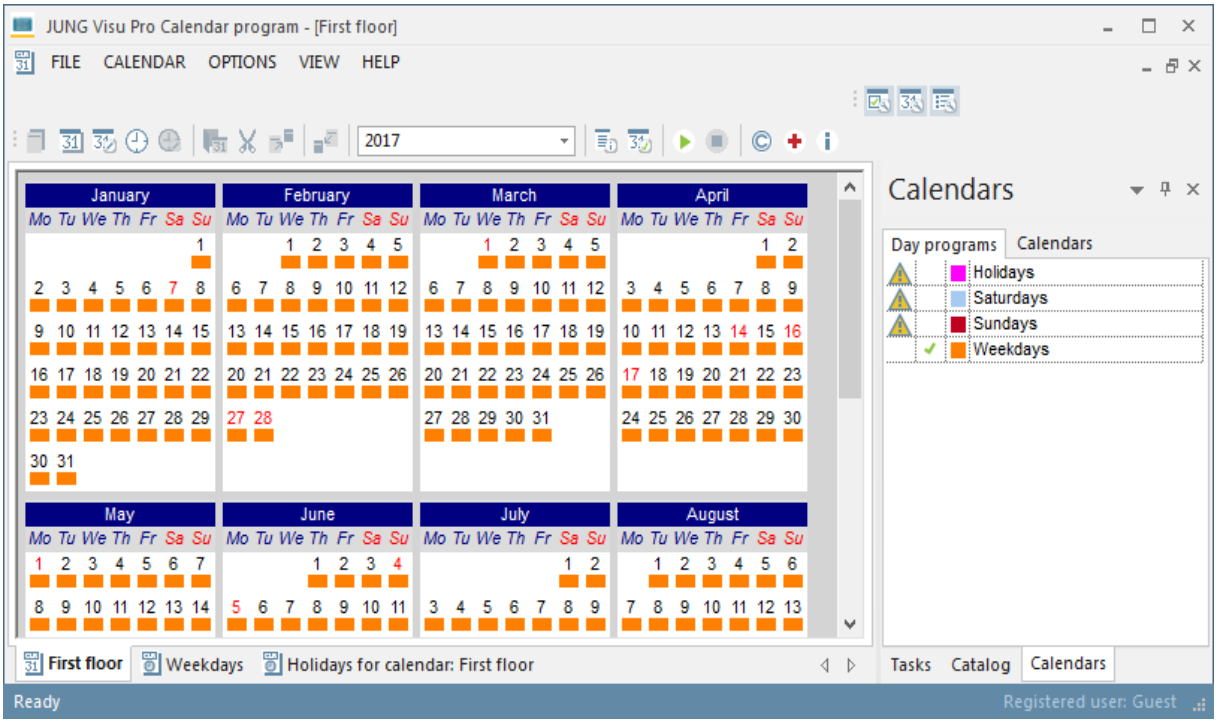
The symbol ✔ in front of the day program name indicates that the day program has been set as standard day.

Note: *If you want to change the standard day, double-click in front of the day program you wish to set as standard day. The standard day is then automatically removed from the day program previously defined as standard day.*

Example: If the day program Saturday *is set as standard day and you want to set the day program* Weekday *as standard day, double-click on the column in front of the day program* Weekday*. The standard day is automatically removed from* Saturday *and assigned to* Weekday*.*



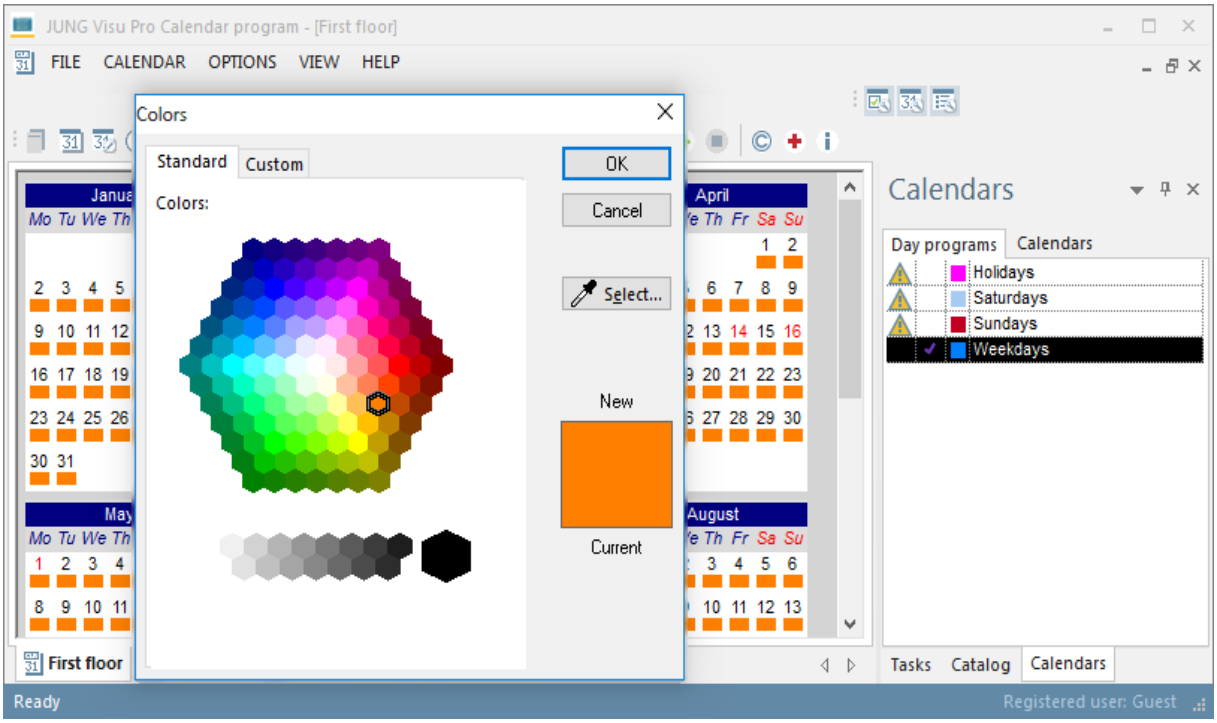
The day program *Weekday* is then automatically assigned to all days in the calendar.



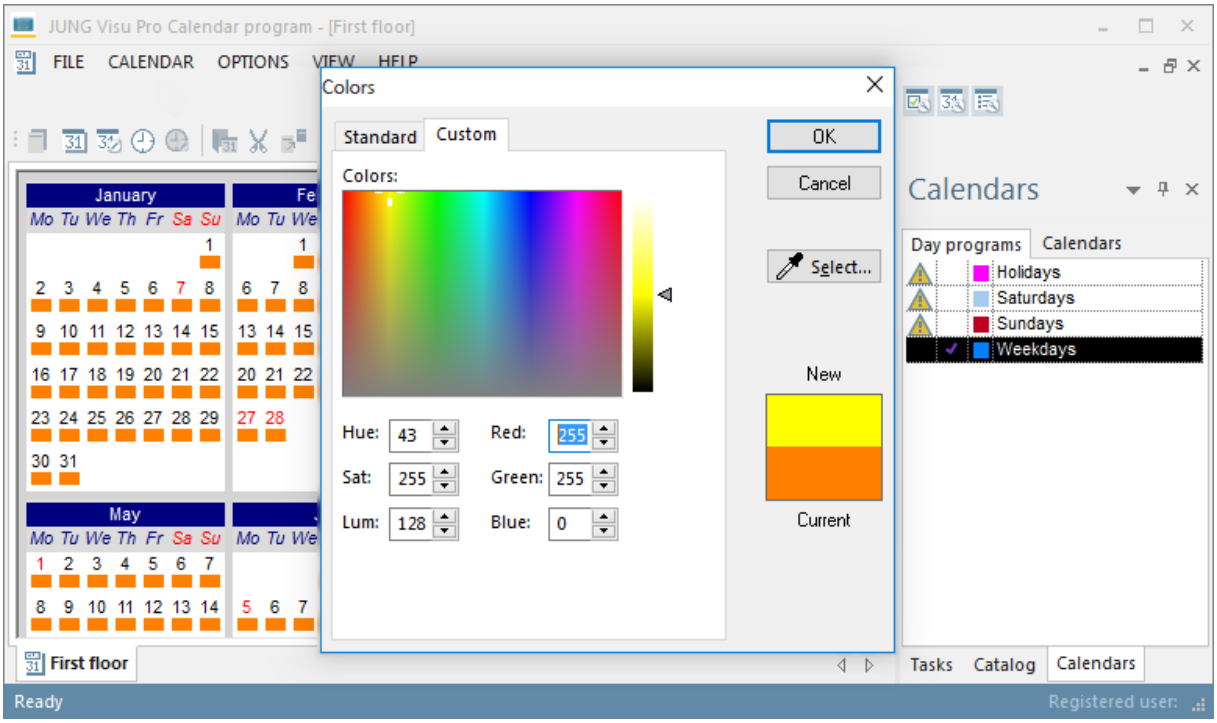
Change color of day program:

To change the color of a day program, choose the tab ***Day programs*** in the Calendars tool window and double-click the color symbol in the third column in front of the day program name. The dialog ***Colors*** opens.

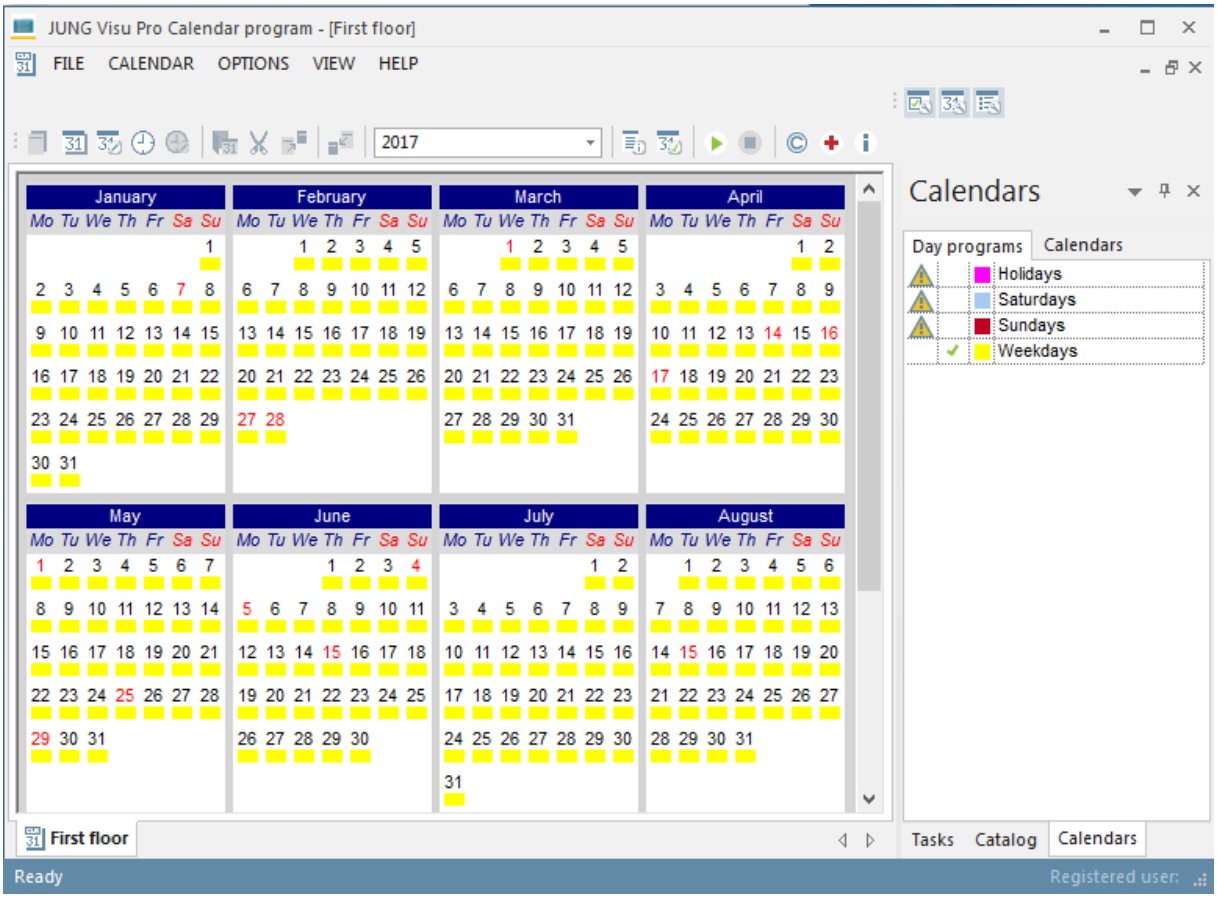
Select the desired color and click ***OK***.



On the tab ***Custom*** you can directly enter the RGB values. To commit the changes, click the button ***OK***.



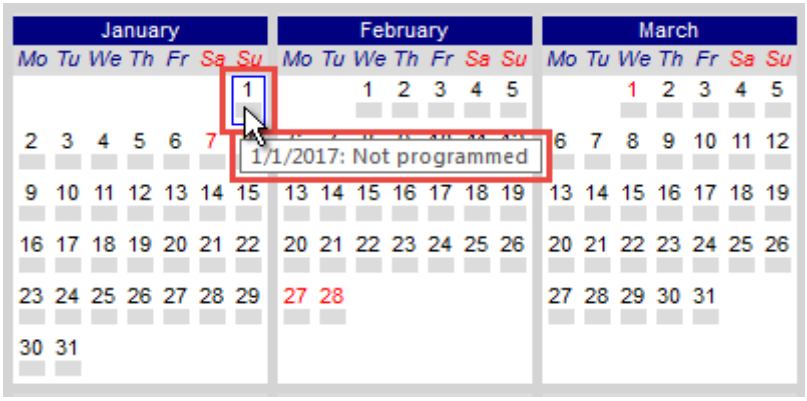
The program color is automatically updated in the calendar.



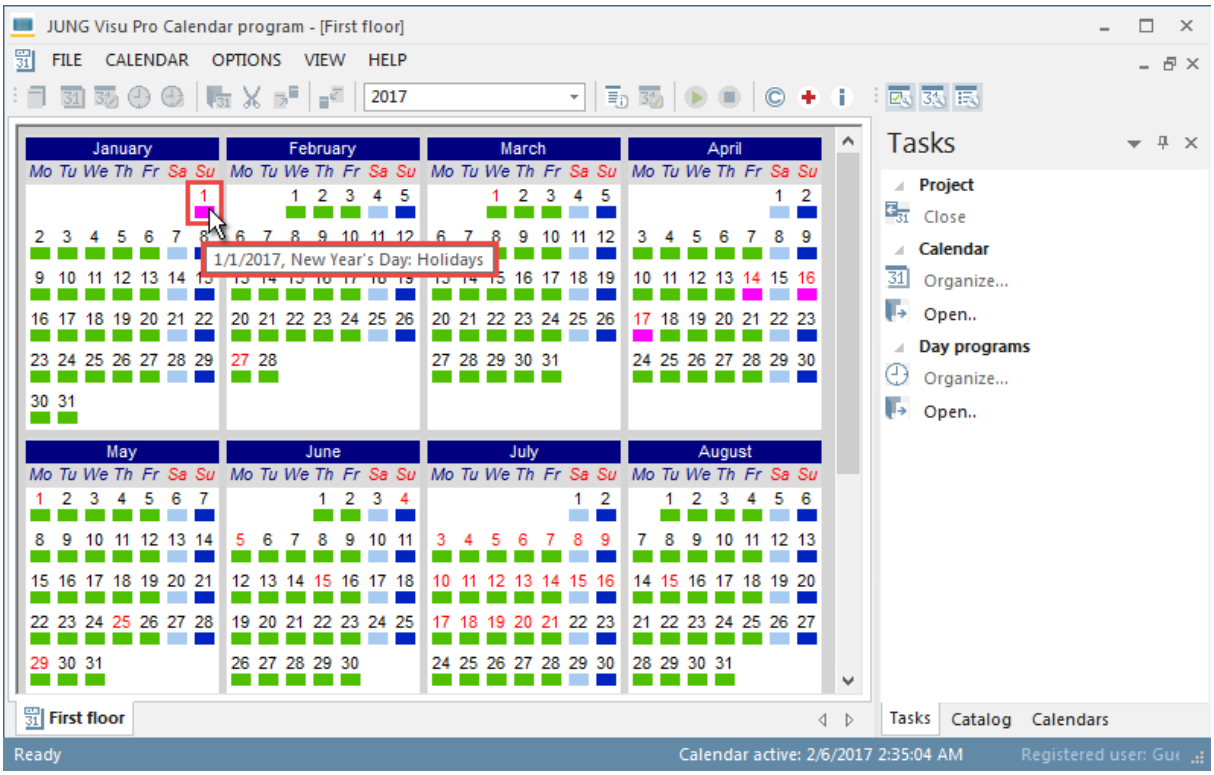
4.4 Calendar Configuration

The calendars comprise a schedule of day programs. The day programs are assigned to calendar days or periods.

The following picture shows a calendar with no day programs assigned. Holidays, other special days and weekends are displayed in red color.



By moving the mouse cursor over a calendar, a tool tip appears showing information about the day the mouse cursor is currently over (date, holiday, name of the day program).



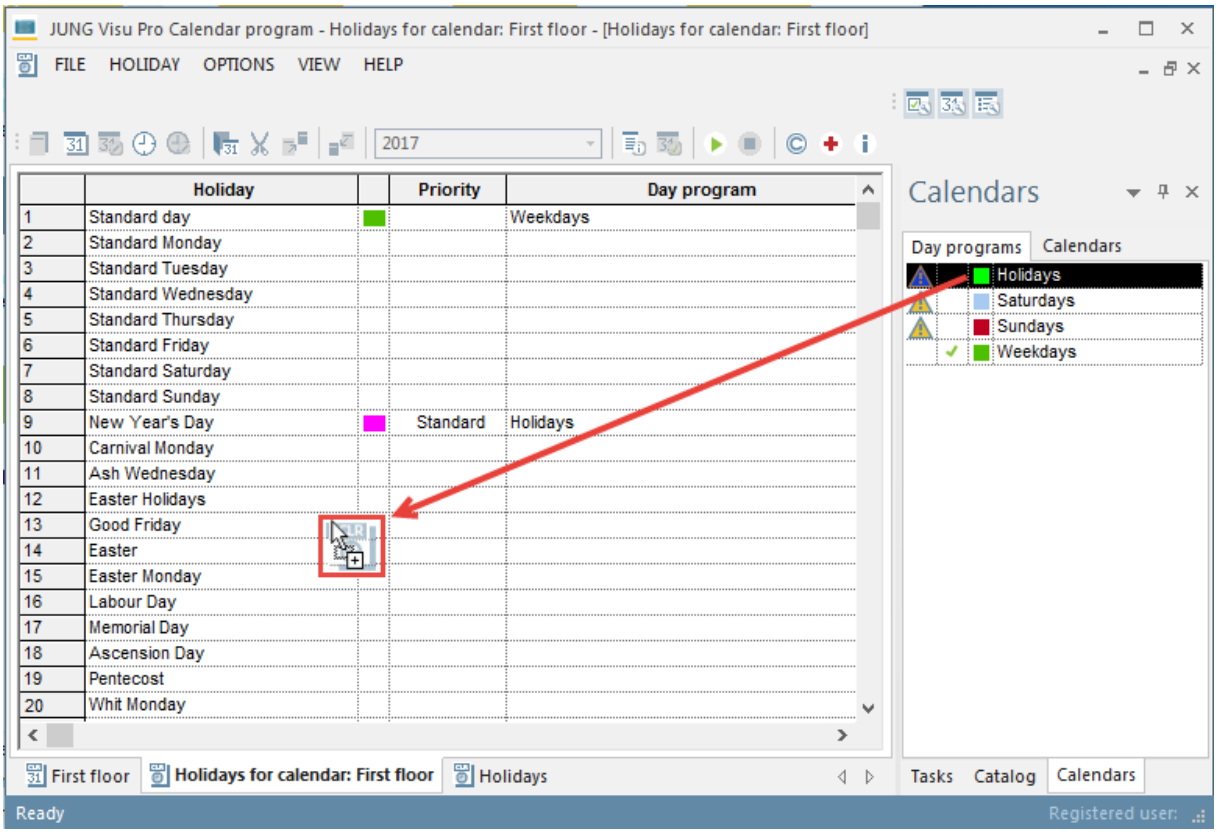
Choose the menu item ***View*** and click ***Holiday programming***. The window ***Holidays for calendar*** is opened where you can define holidays and special days.

4.4.1 Holiday Programming

The window for holiday programming shows a list of predefined special days. Among them is the *standard day*. The day program assigned to the standard day specifies the default commands which are to be performed if no other special day applies.

Assigning day programs to holidays:

To assign a day program to a holiday or special day, select the day program in the tool window ***Day programs*** and drag it to the row of the special day in the holiday programming window.



Then click in the column ***Priority*** to set the priority of the day program.

Priority numbers:

You may assign a day program to Sundays and a different day program to the holiday ***Easter Sunday***, which is a Sunday as well. The priority function is used to resolve such ambiguities.

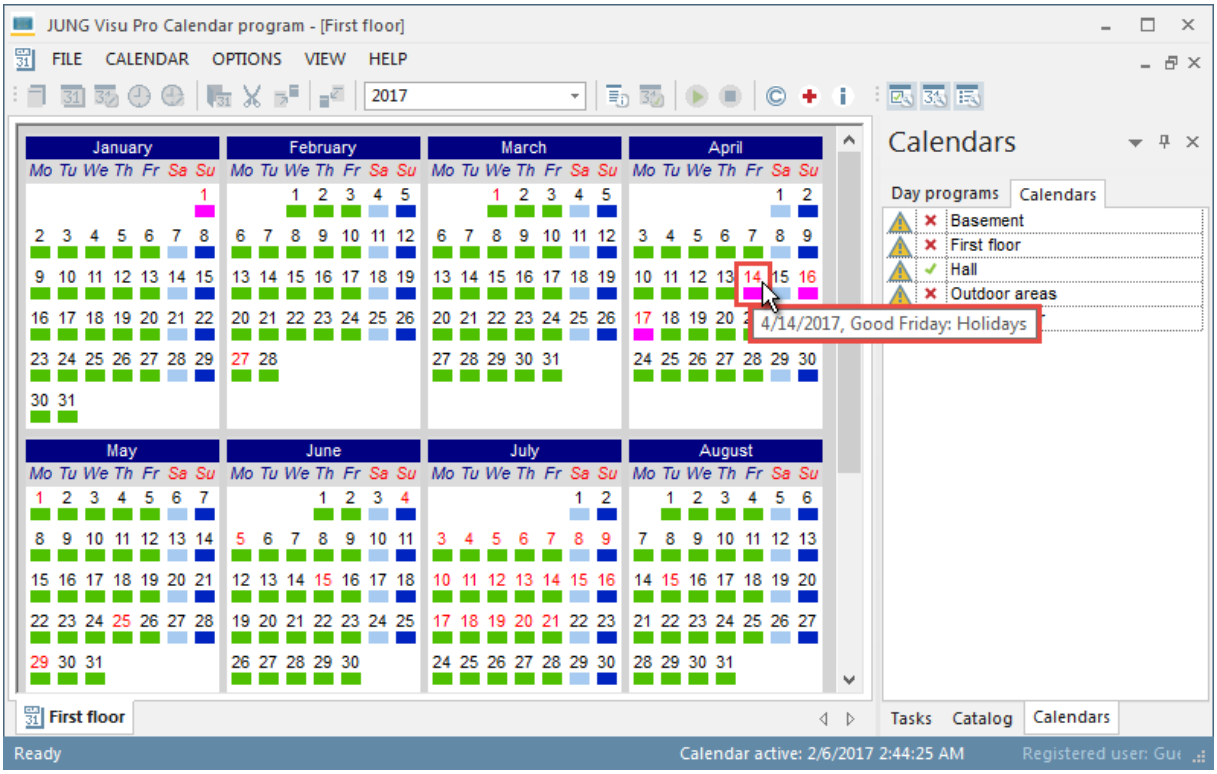
Example: *Easter vacation, Good Friday, Easter Sunday, Easter Monday*

Good Friday, Easter Sunday and Easter Monday are holidays in the Easter vacation. In the example below, day programs are assigned to Good Friday, Easter Sunday and Easter Monday and another day program is assigned to the Easter vacation. If you want the day programs of Good Friday, Easter Sunday and Easter Monday to run instead of the day program of Easter vacation, their priority number must be higher than the priority number of the Easter vacation.

Standard: Lowest priority.

Priority 1-9: The priority rises with the number, 9 being the highest priority.

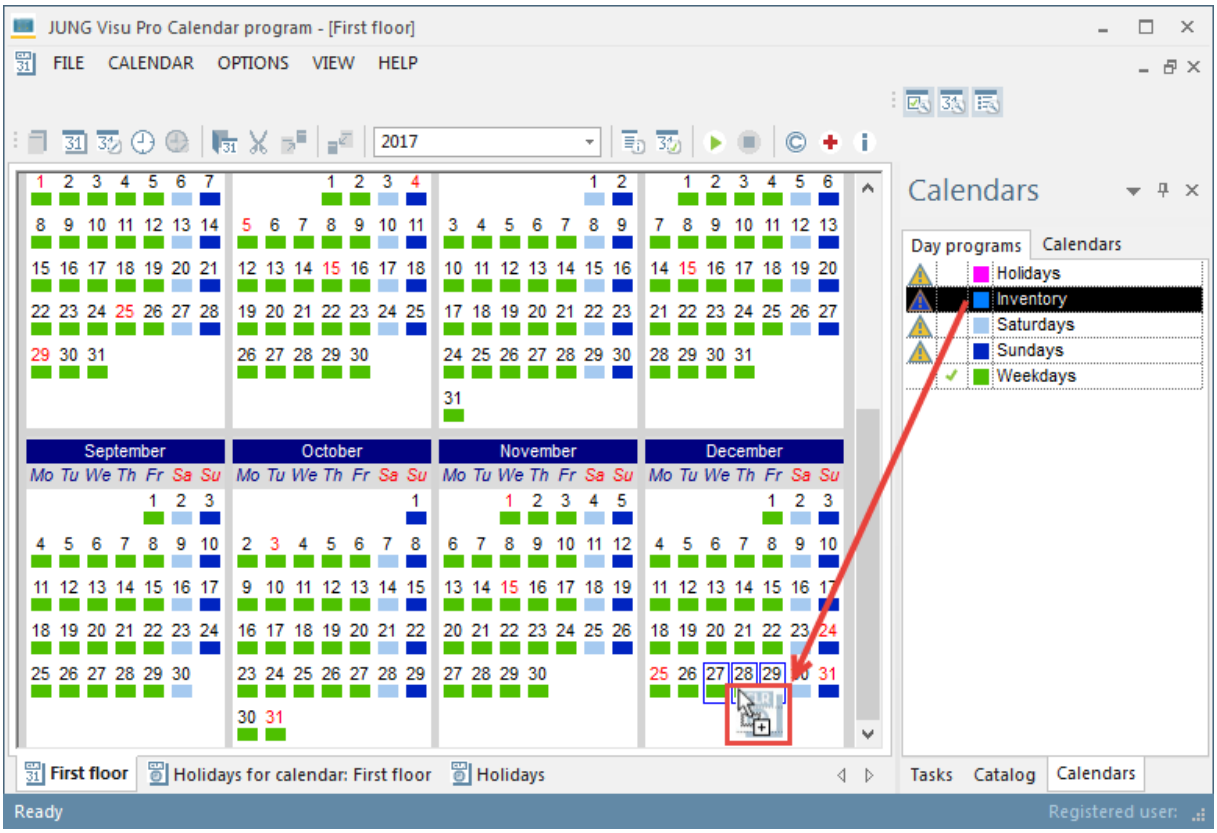
The programming of holidays is automatically updated in the calendar view.



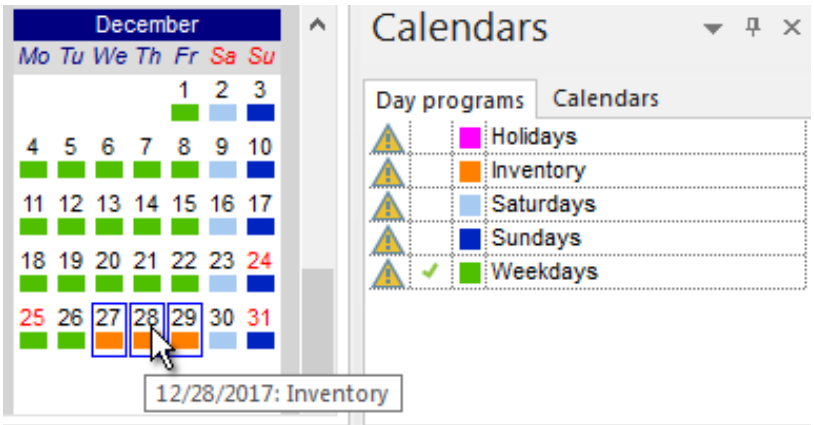
After the holidays have been programmed, the programming of the calendar is completed.

4.4.2 Insert Day Programs Manually

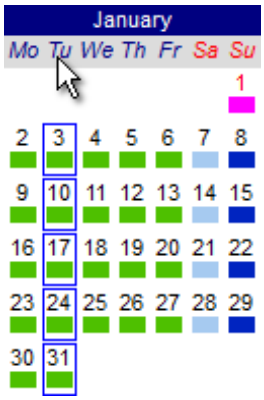
You can assign a day program manually to a day in the calendar. For this purpose, choose the tab ***Day programs*** in the Calendars tool window and drag the desired day program to the desired day in the calendar. If you want to assign the day program to more than one day, select in the calendar the corresponding days you want to assign the day program to. Then drag the day program to the selected days.



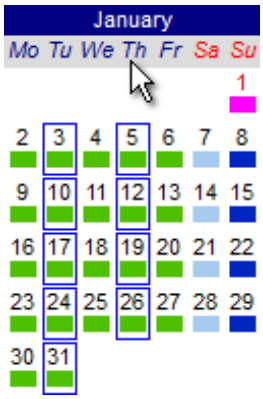
When dropped on the calendar, the day program is assigned to the selected days:



Selection of dates in the calendar:

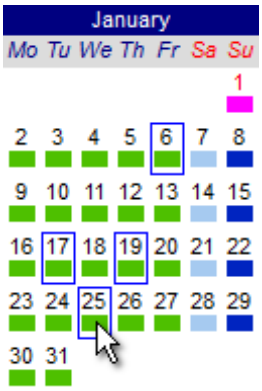


Select a column:
Click on the weekday name.

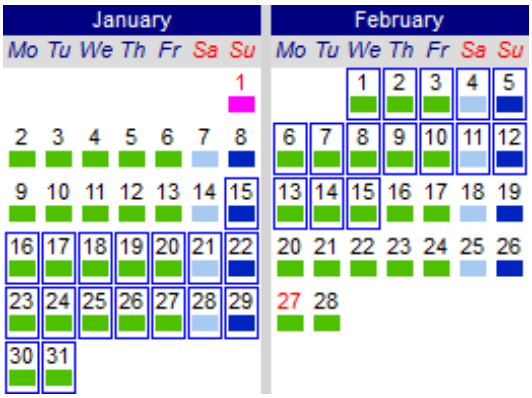


Select multiple columns:
To extend the selection to further columns, hold down the Ctrl key and click on the desired weekday names.

Select a month:
Click on the month name. To extend the selection to further months, hold down the Ctrl key and click the desired month names.



Select several days:
Click on one day, hold down the Ctrl key and extend the selection by clicking on the desired days.



Select a range:
Click on the first day (15.01.), hold down the Shift key and click on the last day (15.02). This method also applies to the selection of a range of months.

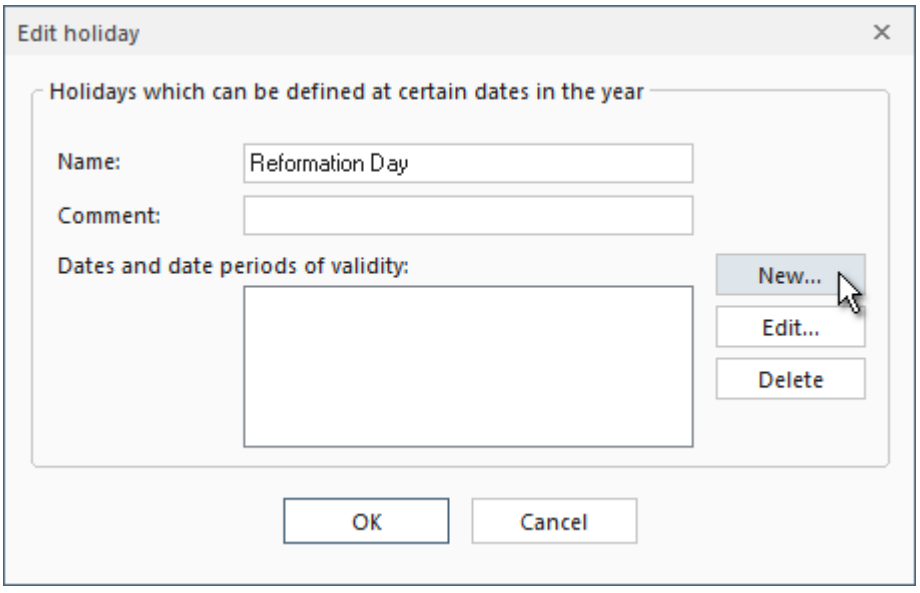
4.5 Holiday Definitions

Choose the menu item ***View*** and click ***Holiday definitions***. The dialog ***Holidays*** opens in the workspace.

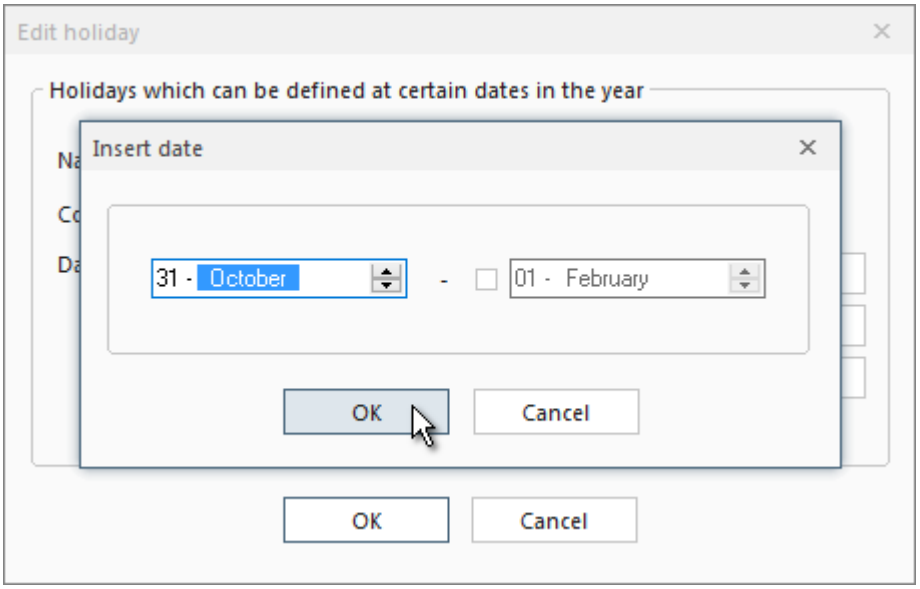
	Holiday name	Description	Comment
1	New Year's Day	1. January	
2	Carnival Monday	48. day before Easter	
3	Ash Wednesday	46. day before Easter	
4	Easter Holidays	4/1/2004 - 4/16/2004, ...	
5	Good Friday	2. day before Easter	
6	Easter	Easter	
7	Easter Monday	1. day after Easter	
8	Labour Day	1. May	
9	Memorial Day	Last Monday in May	
10	Ascension Day	39. day after Easter	
11	Pentecost	49. day after Easter	
12	Whit Monday	50. day after Easter	

4.5.1 Define Annual Holidays as Days of the Year

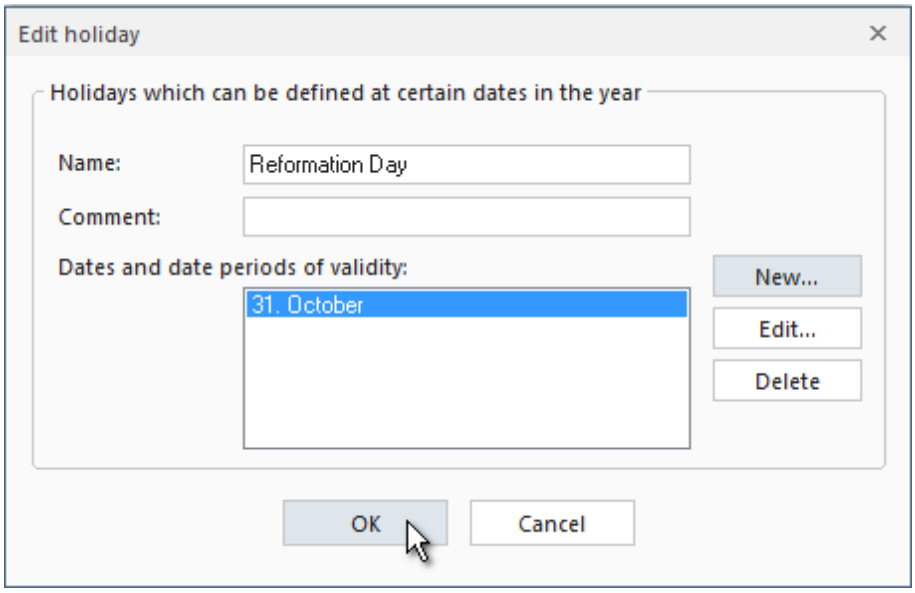
The definition of annual holidays as days of the year allows you to define holidays at certain dates. The entered holiday dates or date periods will be assigned to each year in the calendar.
Choose the menu item ***Holiday*** and the option ***Create*** and click ***as annual holidays as days of the year***. The dialog ***Edit holiday*** opens.



Enter here the name of the holiday. The specification of a comment is optional. Then click the button ***New***. The dialog ***Insert date*** opens.



Specify the date of validity and click the button ***OK***.



To edit a date, select in the dialog ***Edit holiday*** the date and click the button ***Edit***. The dialog ***Insert date*** opens where you can change the date. Click ***OK*** to commit the change.

To delete a date, select the date and press the button ***Delete***.

To complete the editing of the holiday and to commit the entries, click the button ***OK***.

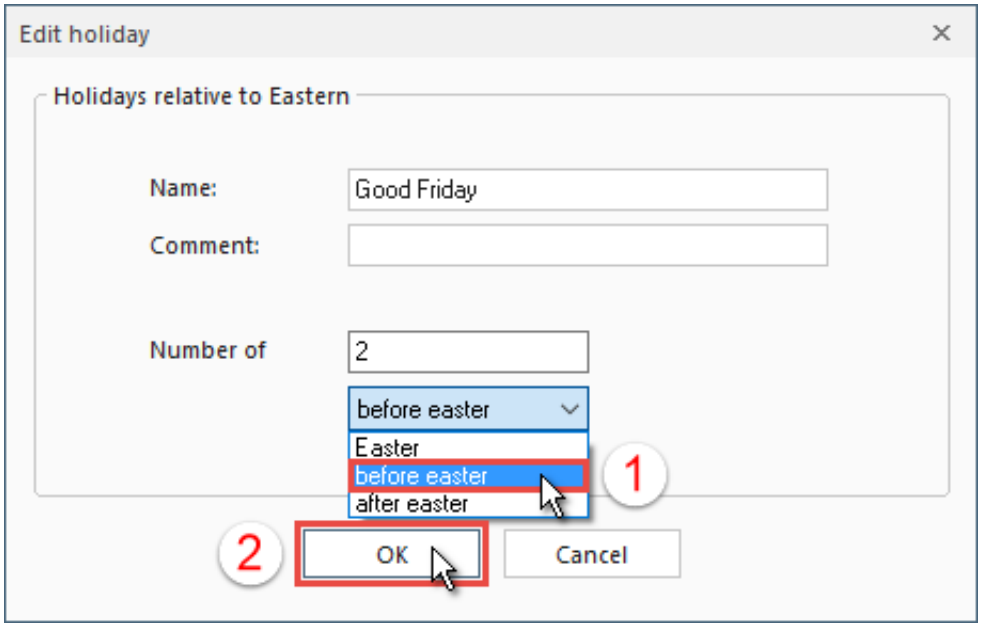
4.5.2 Define Holiday Relative to Easter

Many holidays do not take place on a fixed day in the year, but relate to Easter which itself takes place on varying calendar dates.

Choose the menu item ***Holiday*** and then ***Create*** and click ***relative to Easter***. The dialog ***Edit holiday*** opens where you can enter the name of the holiday. The specification of a comment is optional.

Then enter the number of days before or after Easter. Good Friday, for instance, takes place two days before Easter. If the holiday you want to define is Easter Sunday, choose the option ***Easter***.

The calendar will calculate the holiday for each year.



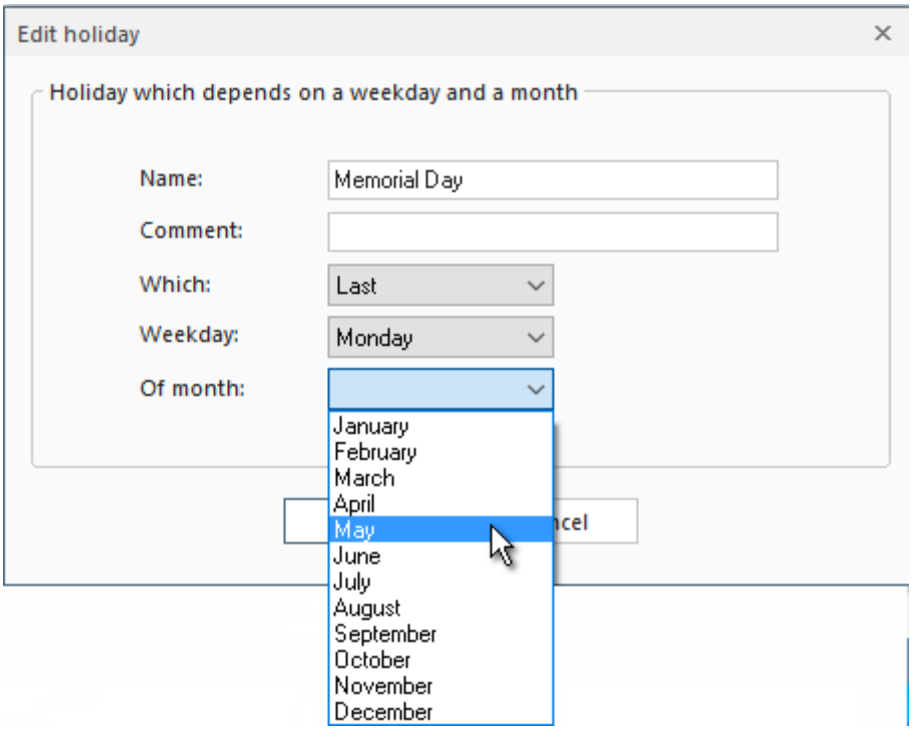
To commit the entries, click the button ***OK***.

4.5.3 Define Holiday as Weekday in a Month

Choose the menu item ***Holiday*** and then ***Create*** and click ***as weekday in a month***. The dialog ***Edit holiday*** opens where you can enter a holiday which depends on a weekday and a month.

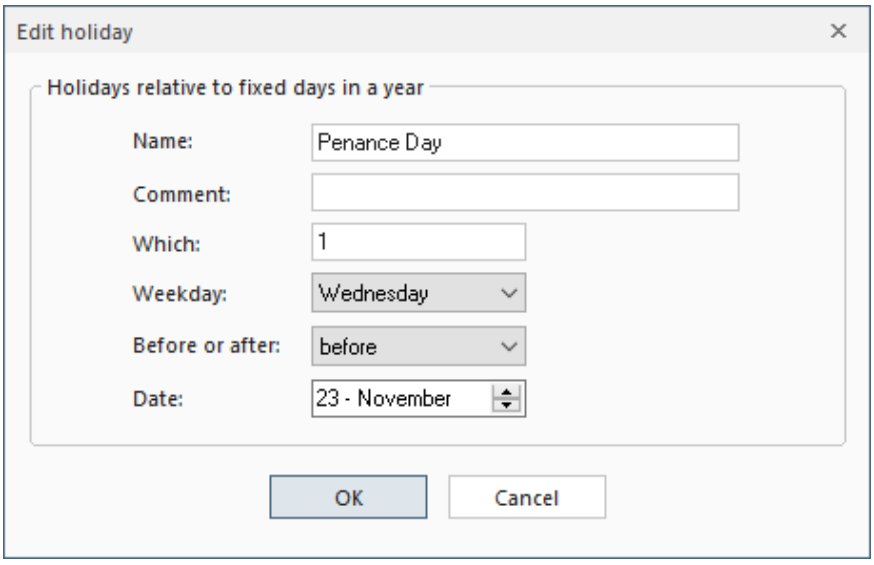
You can define that the first/second/../last weekday of a particular month should be the holiday or special day. Then specify on which day of the week the holiday should take place and in which month.

The calendar will calculate the holiday for each year.



4.5.4 Define Holiday Relative to Fixed Day of the Year

Choose the menu item ***Holiday*** and then ***Create*** and click ***relative to fixed day of the year***. The dialog ***Edit holiday*** opens.

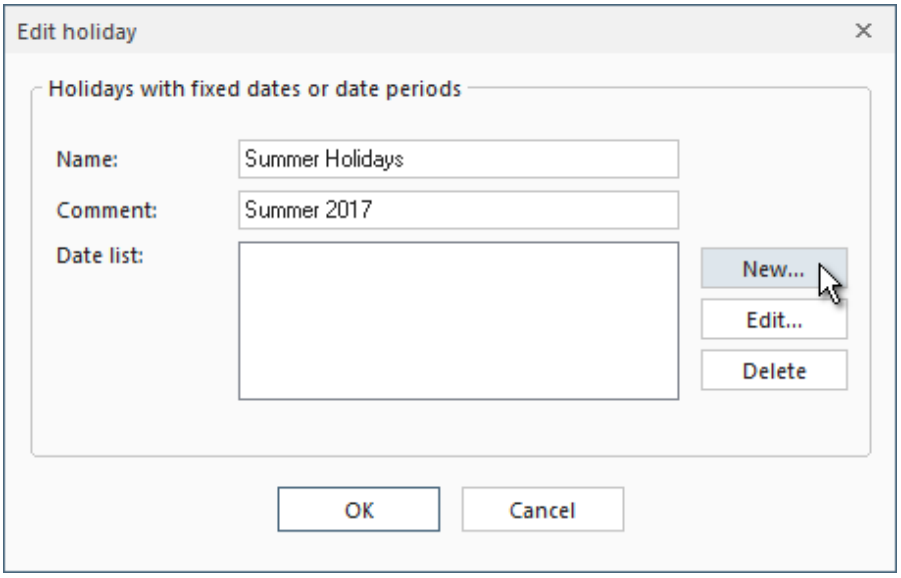


Enter the name of the holiday. The specification of a comment is optional. Then enter on which weekday of the month the holiday should take place (first – fourth or last), which weekday it should be and if it should take place before or after the reference date.

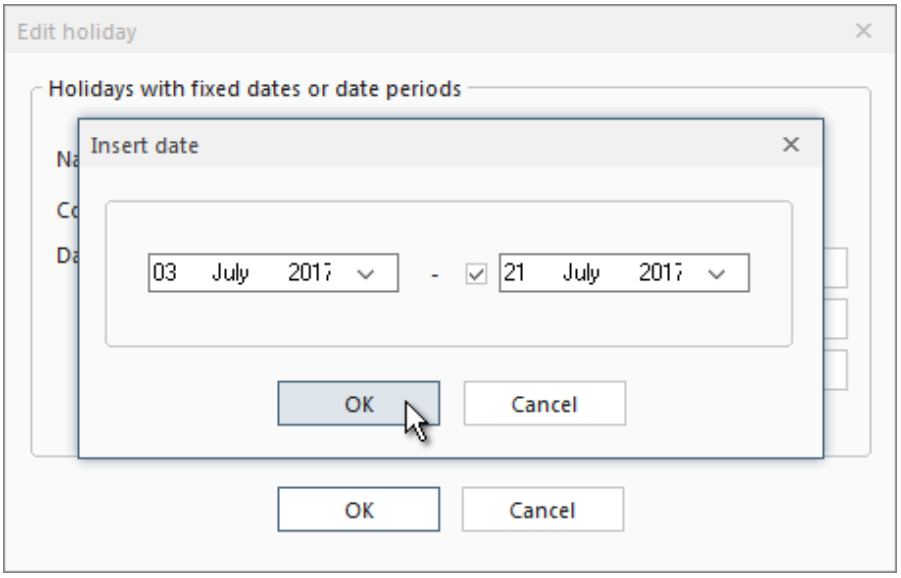
To commit the entries, click the button ***OK***. The calendar will calculate the holiday's date for each year.

4.5.5 Define Holidays with Fixed Dates

You can define a time period (for example school holidays, vacation and so on) for a year or several years in advance. Click the menu item ***Holiday*** and then ***Create*** and click ***as holidays with fixed dates***. The dialog ***Edit holiday*** opens where you can enter a holiday with fixed dates or periods of time.



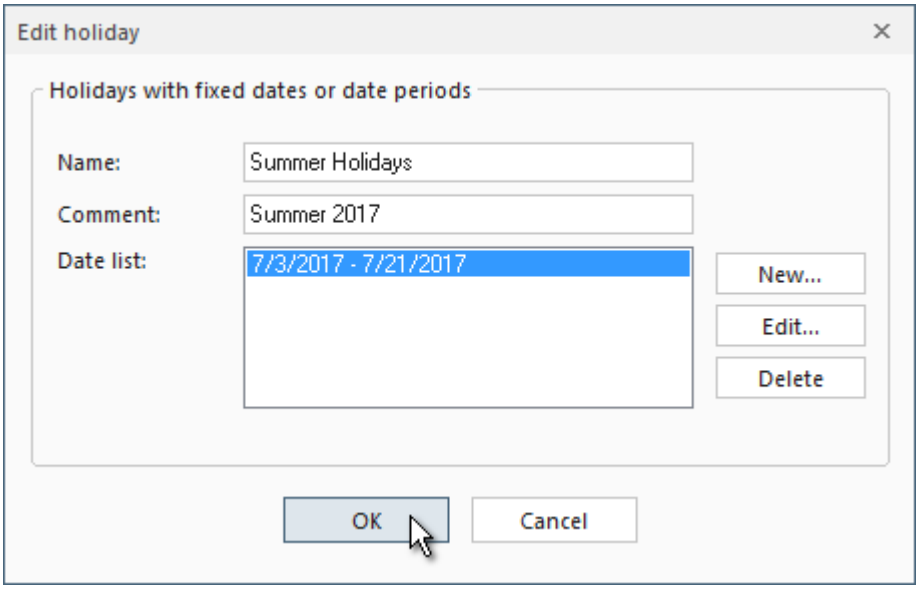
Click the button ***New***. The dialog ***Insert date*** opens.



Enter the date for the holiday in the first entry field. If you want to define a period of time, enter the start date of the holiday, then select the checkbox and enter the end time. Click the button ***OK*** to commit the entries.

To edit a date, select in the dialog ***Edit holiday*** the date and click the button ***Edit***. The dialog ***Insert date*** opens where you can change the date. Click ***OK*** to commit the change.

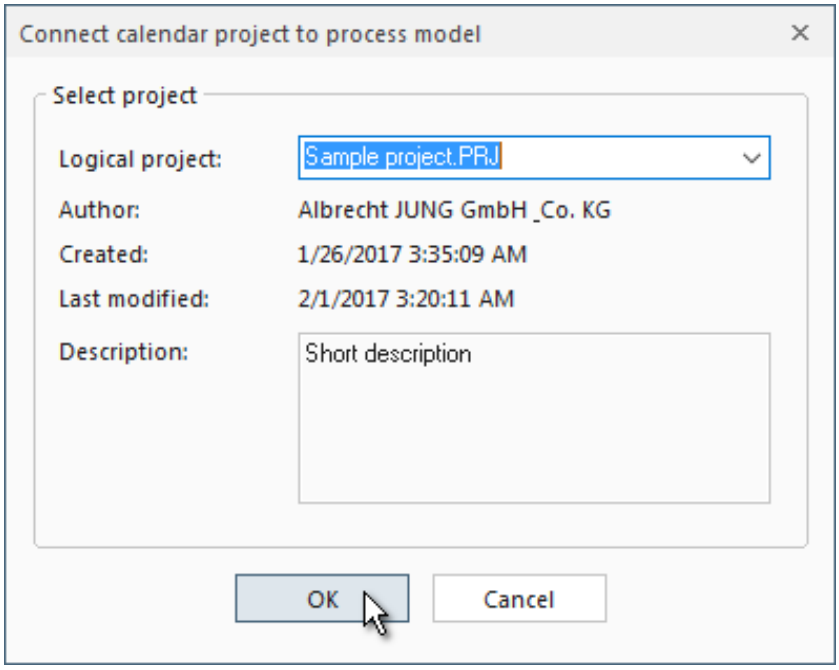
To delete a date, select the date and press the button ***Delete***.



To complete the editing of the holiday and to commit the entries, click the button ***OK***. In the dialog ***Edit holiday*** click the button ***OK*** to complete the definition of the holiday.

4.6 The Process Model

A calendar project must be connected to a process model, which is the target of the day commands. You can change the logical project connected to the calendar project: choose the menu item ***File*** and click ***Select project***. The dialog ***Select project*** opens where you can select another process model.



Select the process model and press the button ***OK*** to confirm the selected connection to the process model and to close the dialog.

Update process model:

> ***Important:***
>
> *If the process model has been changed, the calendar project must update the definitions of process variables.*

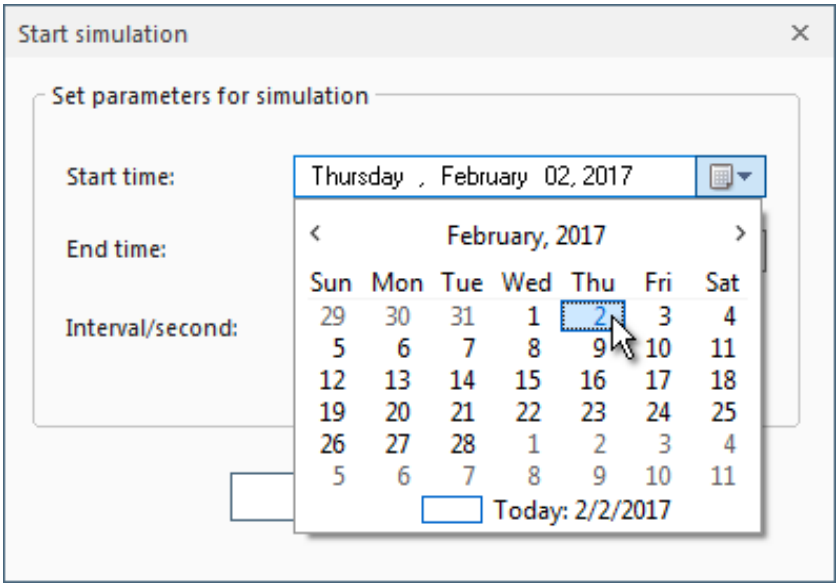
Choose the menu item ***File*** and click ***Update project*** or the function key ***F5*** to update the modified data of the process model in the calendar project. Then check the validity of the day programs.

4.7 Simulation

The calendar application provides a simulation function which serves to display the effects of the planning in fast motion.

Start simulation:

Choose the menu item ***Options*** and click ***Start simulation***. The dialog ***Start simulation*** opens where you can specify simulation properties.



Enter the start and end time of the simulation and set a scaling for the time schedule.

Press the button ***OK*** to start the simulation.

The simulation state will be indicated in the status bar.

The messages list indicates the commands that normally would be sent to the process model.

Stop simulation:

To stop the simulation, choose the menu item ***Options*** and click ***Stop*** or the button in the toolbar.

4.8 Applying the Calendar Project

Start calendar project:

To start the time schedule of the calendar project, choose the menu item ***Options*** and click ***Start*** or click the button in the toolbar.

The running of the time schedule will be indicated in the status bar.

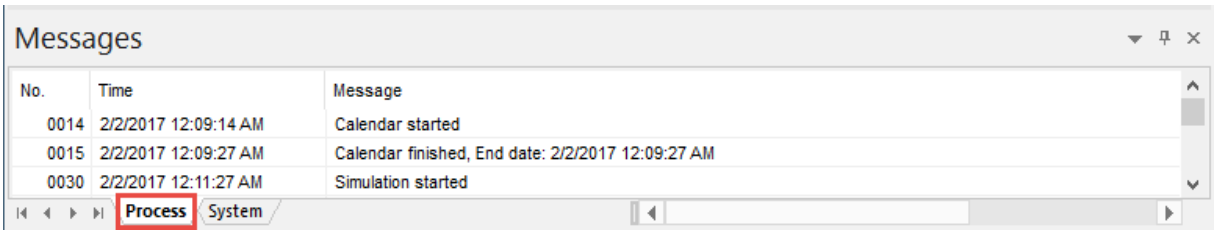
The KNX editor starts automatically and connects to the bus if the option ***Auto connect when project is opened*** has been enabled in the dialog ***KNX project settings***.

To stop the time schedule, choose the menu item ***Options*** and click ***Stop*** or click the button in the toolbar.

5 Message List

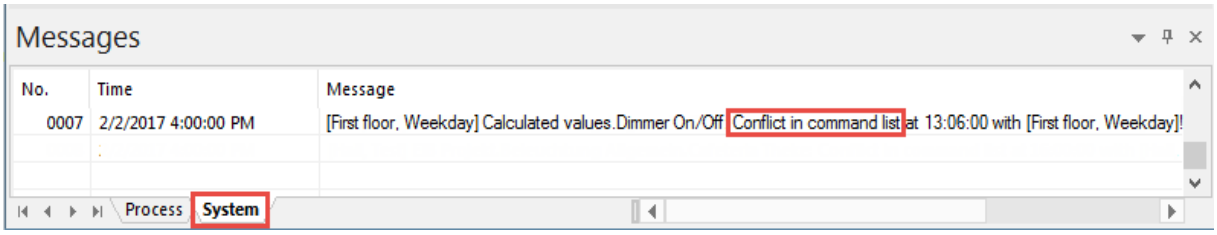
The message list shows process and system messages for the calendar project or the simulation. It can be shown or hidden at the bottom of the calendar program editor.

Process messages:



The tab ***Process*** displays operations issued by the time schedule of the calendar project.

System messages:



The tab ***System*** displays conflicts of the time schedule. A conflict occurs if two commands are sent to one process variable at the same time.

Show messages: Choose the menu option ***View*** and click ***Messages*** to toggle the visibility of the message list.

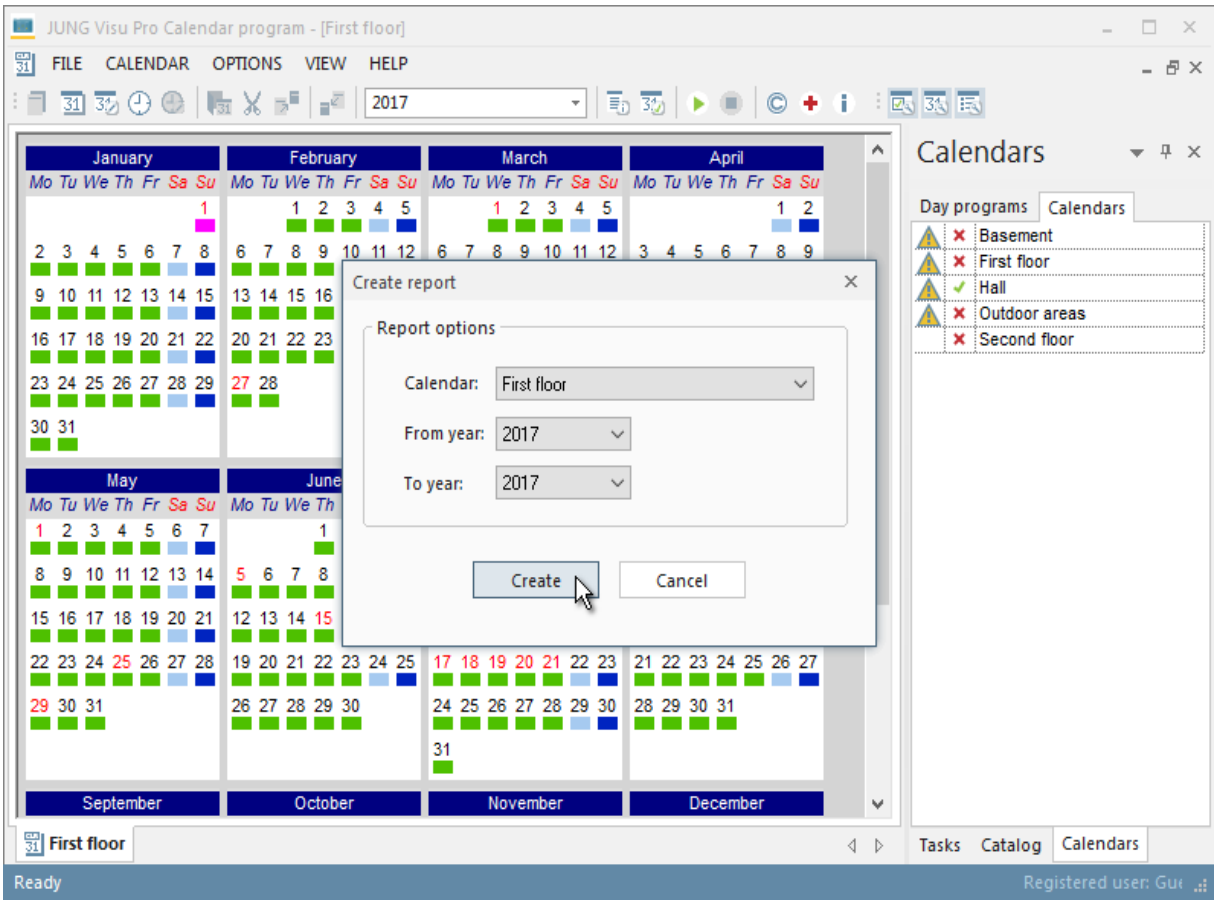
Clear messages: Choose the menu item ***View*** and click ***Delete messages*** to delete the messages in the message list.

6 Reports

- Reports provide overviews over the project, including calendars, day programs, holidays and connections to a process model.
- Reports can be printed (with print preview).

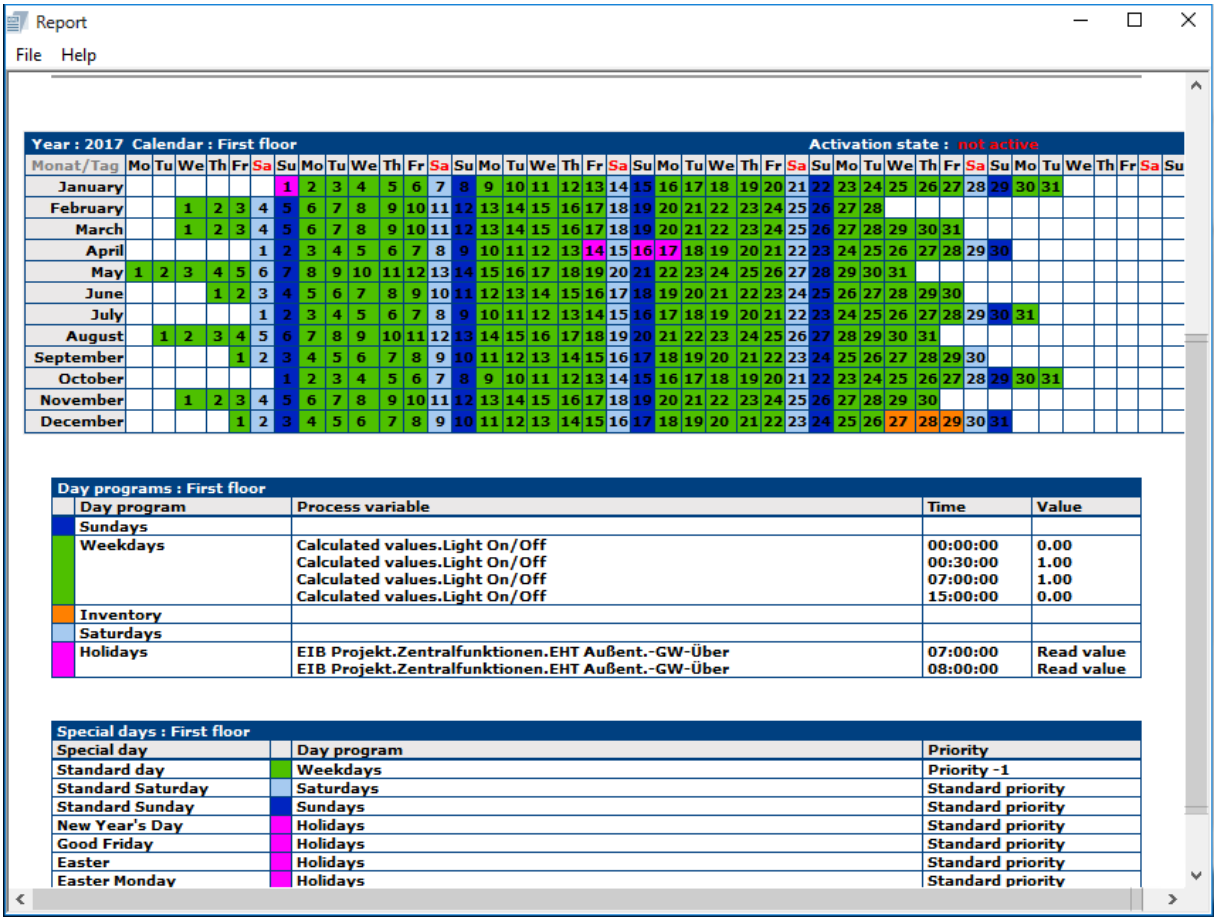
6.1 Create Report

To create a report for the current project, choose the menu item ***File*** and click ***Report***. The dialog ***Create report*** opens. Select the desired options for the report generation and click the button ***Create***.

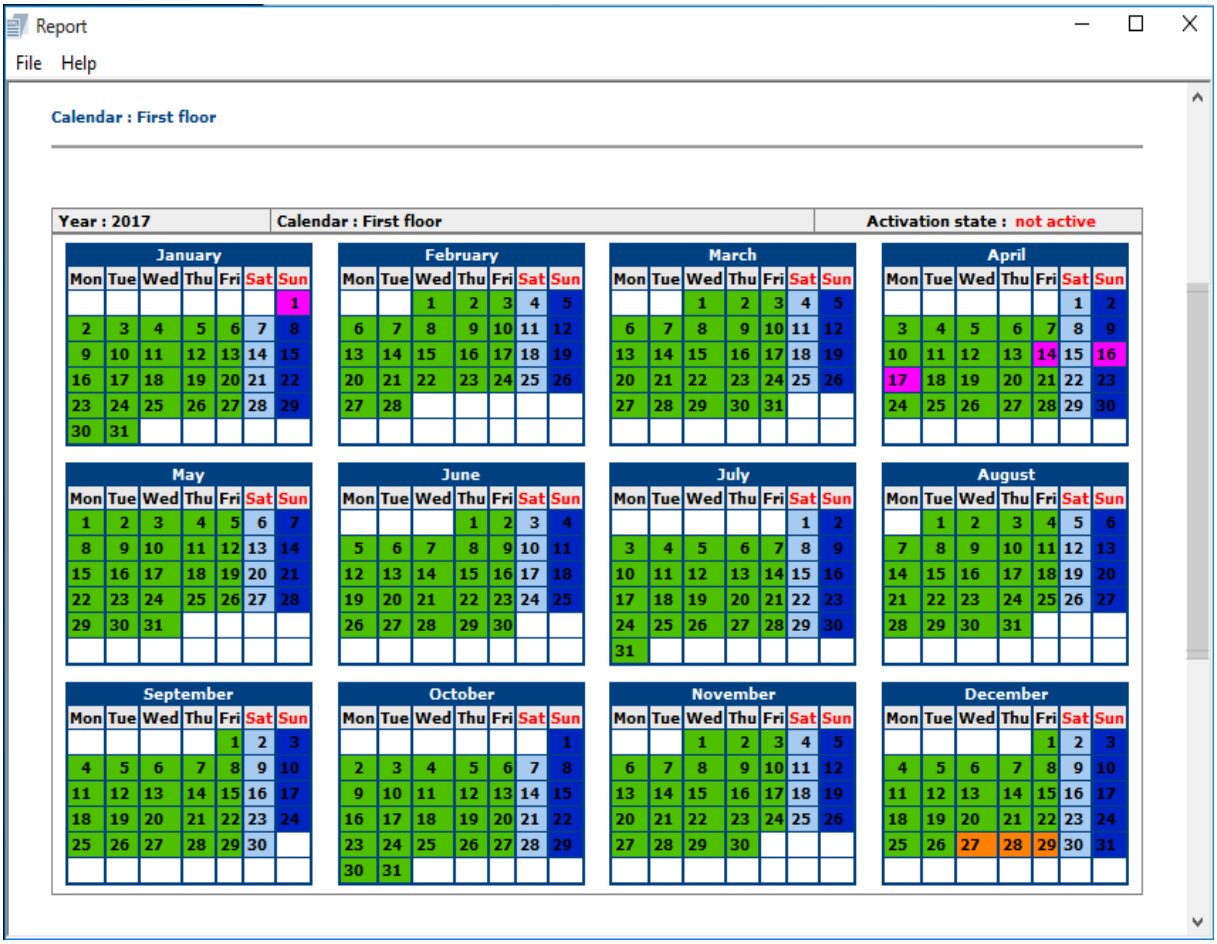


The report provides different views.

Year calendar: Shows the calendars for selected years in a compact format along with day programs.



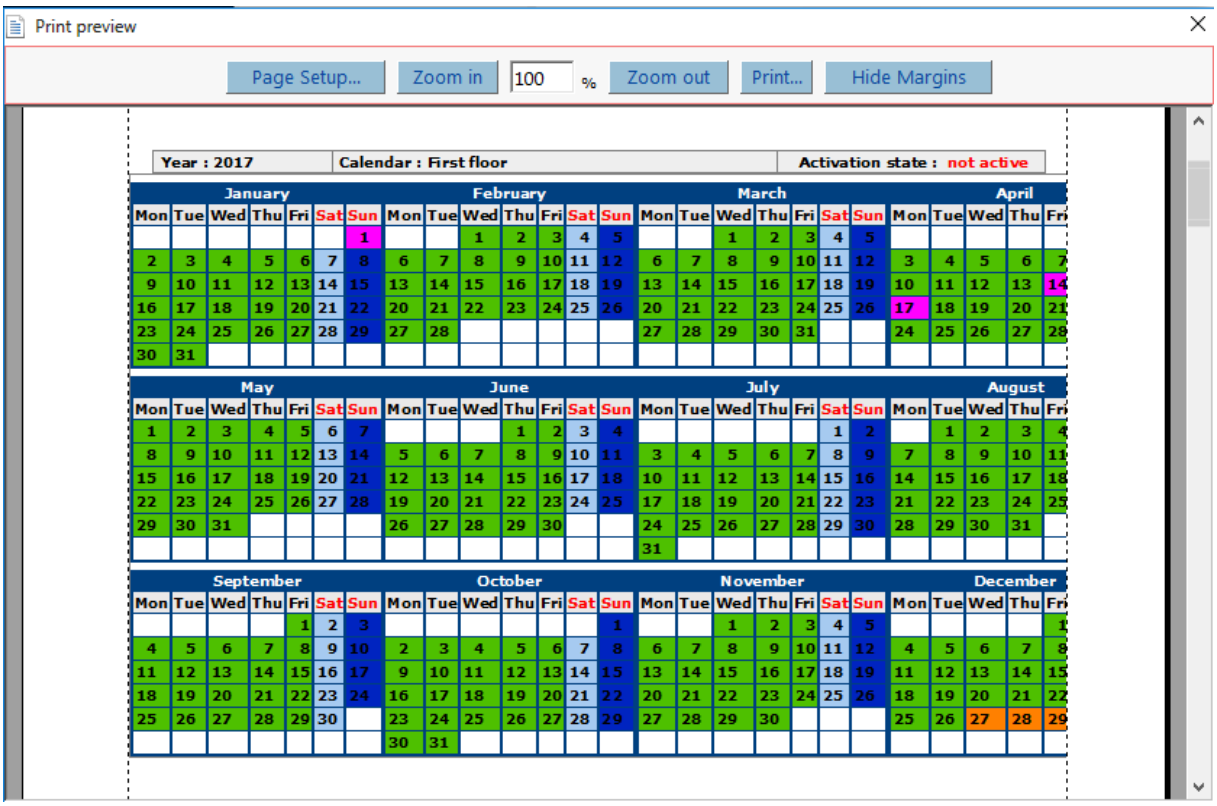
Month calendars: This is an alternative view of the year.



Printer friendly: Calendars are displayed in black and white.

Special day: Displays the definitions of holidays and other special days.

6.2 Print Preview



The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these ↦,↧ markers the user has visual control over the margin settings.

JUNG Visu Pro
XPL Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation systems....3
1.3 Squeezebox Server....3

2 INTRODUCTION....4

2.1 XPL Editor: Control Streaming of Digital Music....4
2.2 First Steps....5
2.3 Placing into Operation....6

3 USER INTERFACE....7

3.1 Menu....7
3.2 Toolbar....9

4 FUNCTIONS....9

4.1 Project Organizer....9
4.2 Squeezebox....14
4.3 Device Settings....17
4.4 Playlists and Textlists....19
4.5 Message List....20

5 REPORTS....21

5.1 Create Report....21
5.2 Print Preview....23

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 Bit per Pixel.
Squeezebox – devices from the company Slim Devices, Inc.	

1.2 Operation systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Squeezebox Server

Using streaming of digital music requires the installation of the Squeezebox Server software on a second PC. The Squeezebox Server software is an open source and can be downloaded from the Internet.

2 Introduction

2.1 XPL Editor: Control Streaming of Digital Music

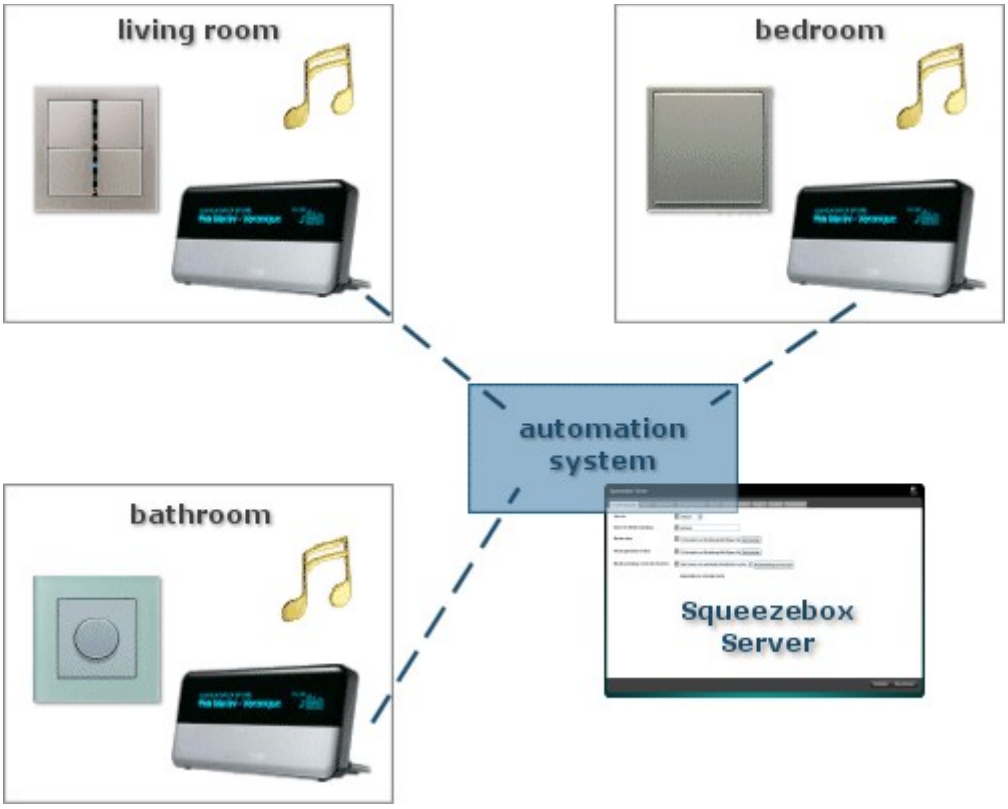
There is an increasing desire to combine home and building automation with the streaming of digital music.

For instance lighting and music control can be integrated in scenes which can be selected at the push of a button from anywhere in the building.

One of the best systems for audio streaming over IP networks is the Squeezebox – a system from the company Slim Devices, Inc. Audio streams are transmitted over Ethernet or wireless (IEEE 802.11) networks from a server with the open-source Squeezebox Server software to Squeezebox Music Players, and in turn controlled by the XPL editor of the visualization system.

Supported audio streams are for instance Internet Radio, MP3, WAV, WMA and Ogg Vorbis.

We strongly recommend not to install the Squeezebox Server software at the PC running the visualization system, since the Squeezebox Server could impair the proper operation of the automation system.



The XPL editor of the visualization system controls the Squeezebox Server and in turn the Squeezebox Music Players via the XPL data protocol, which is standardized by the XPL project (see links below).

Among other possibilities, using the XPL editor, push buttons and dimmers can be used to control volume, playlists etc., or messages can be displayed in the display of the music player.

Further information:

http://www.logitech.com	Homepage of Logitech. This is the place to get more information about Squeezebox devices.
http://xplproject.org.uk/	Homepage of the 'XPL' project.
http://www.mySqueezebox.com/download	Here a free software of the Squeezebox Server Software can be downloaded.

2.2 First Steps

In order to clearly distinguish between the different applications, different colors were used. The icon of the **XPL editor** contains a blue colored bar .

The **XPL editor** is part of the visualization package and is used to control streaming of digital music. It is used to create XPL projects and connects to the Squeezebox Server software, which in turn connects to Squeezebox devices.

After the installation of the visualization system, a program group is created in the Start Menu. The standard options can be adopted or changed manually during installation. You may start the control panel application of the visualization system from the WINDOWS program manager. To start an application, select the list entry ***Function Modules*** in the sidebar of the control panel and double-click the icon of the application you wish to start.

Important first steps:

1. Create an XPL project.
2. Create a Squeezebox.
3. Edit the device settings.

For a more detailed description of the individual steps, see chapter ***4 Functions***.

Requirements: *The Squeezebox Server software must be installed.*

For each Squeezebox device the Squeezebox Server software defines a player name, which can be edited via the Squeezebox Server settings menu. The player name is used as the device's XPL identification.

2.3 Placing into Operation

For placing into operation the Squeezebox devices and the Squeezebox Server, please read the documentation of the respective manufacturers, available in the Internet. We do not provide support for Squeezebox devices or Squeezebox Server software.

We strongly recommend to install the Squeezebox Server software on a second PC and not on the PC running the visualization system.

To connect the XPL editor with the Squeezebox Server software, the Squeezebox Server must be configured accordingly. You will find respective settings in the menu ***Server settings*** **>** ***Network*** **>** ***XPL*** (or similar) of the Squeezebox Server software.

XPL support for Squeezebox Server must be activated.

On the PC running the Squeezebox Server software, an "XPL – Hub" must be installed. You may download a free XPL-Hub from http://www.xplmonkey.com/xplhub.html.

An XPL-Hub distributes messages from members in the XPL network to applications running on the same PC running the XPL-Hub. The XPL-Hub is required for the Squeezebox Server but not for the visualization system.

The XPL editor does not run with an XPL-Hub unless the WINDOWS registry key

"HKEY_CURRENT_USER\Software\ESF Software GmbH\EsfA\EsfXplEditor\XplSettings\EnableHubMode"

has the value 1 (default is 0).

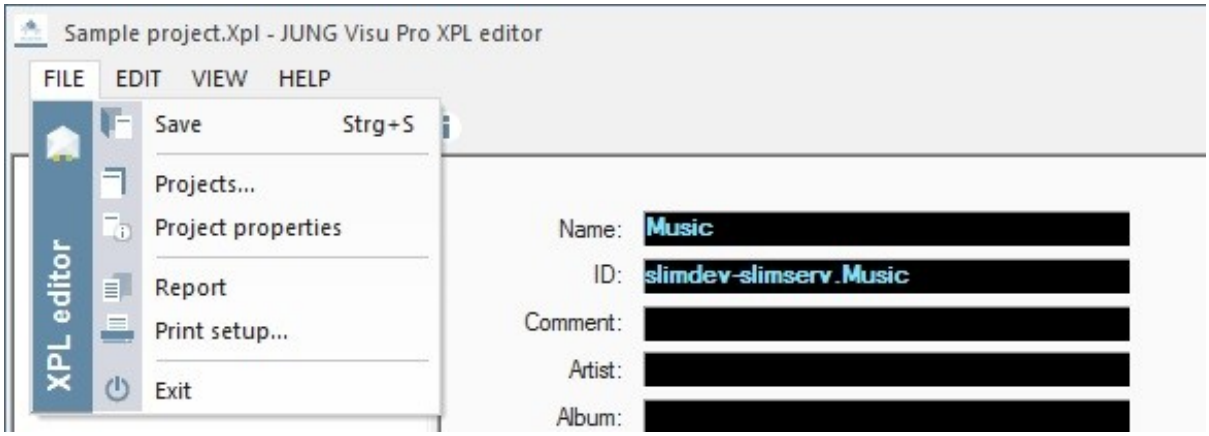
If this key is set to 1, the visualization system and the Squeezebox Server can theoretically run at the same PC, but we strongly recommend to use this option only for testing. Squeezebox Server (and possible plugins to Squeezebox Server) could impair the proper operation of the visualization system.

When using the XPL-Editor and the XPL protocol, you need to be aware of general network issues. In particular you may need to check your firewall settings to allow passing XPL telegrams (TCP/IP, usually port 3865 for receiving).

3 User Interface

3.1 Menu

Menu item *File*



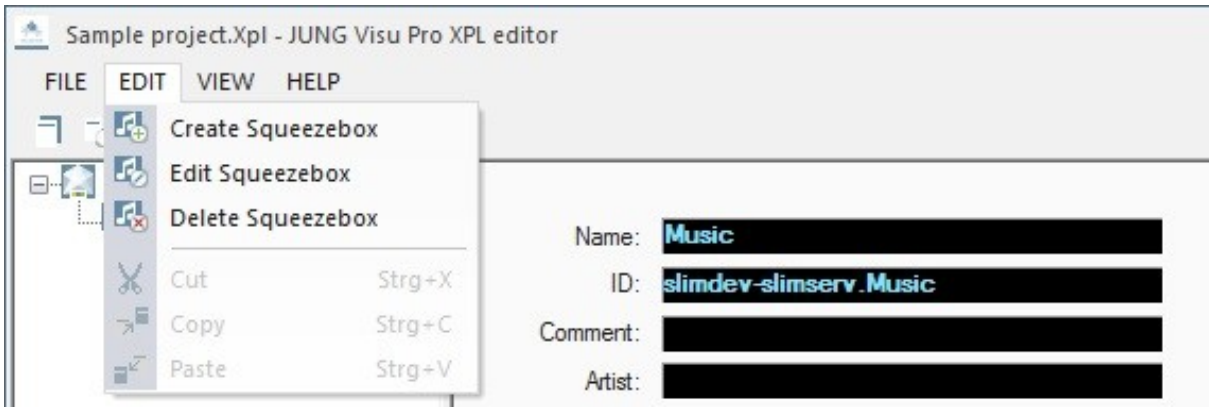
Save: Saves the current project.

Projects: Opens the project organizer dialog. In this dialog projects can be created and maintained.
Project properties: Opens a dialog to define or update general project properties.

Report: Create or view report of the current project.
Print setup: Select printer and printer properties.

Exit: Quits the XPL editor. Shortcut ALT+F4.

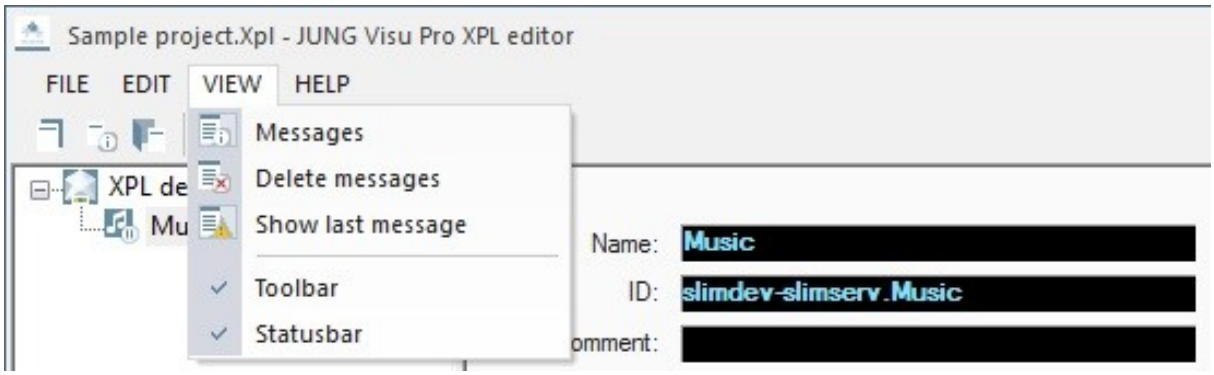
Menu item *Edit*



Create Squeezebox: Opens the dialog ***Create Squeezebox***.
Edit Squeezebox: Opens the dialog ***Edit Squeezebox***.
Delete Squeezebox: Deletes the selected Squeezebox.

Edit: Standard options: Cut, Copy, Paste.

Menu item *View*



Messages: Toggles the visibility of the message list at the bottom of the editor.

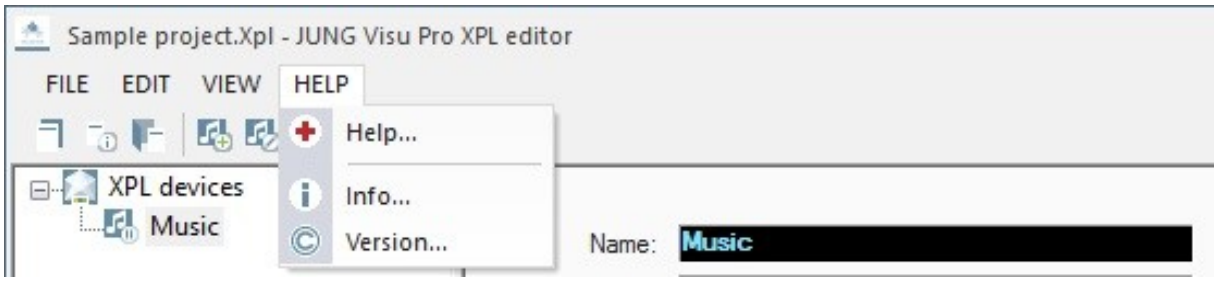
Delete messages: The messages in the message window are deleted.

Show last message: Toggles auto scrolling of the message list. If it is activated and a new message arrives, the message list automatically scrolls to the newly inserted message.

Toolbar: Toggles the visibility of the toolbar.

Statusbar: Toggles the visibility of the statusbar.

Menu item *Help*

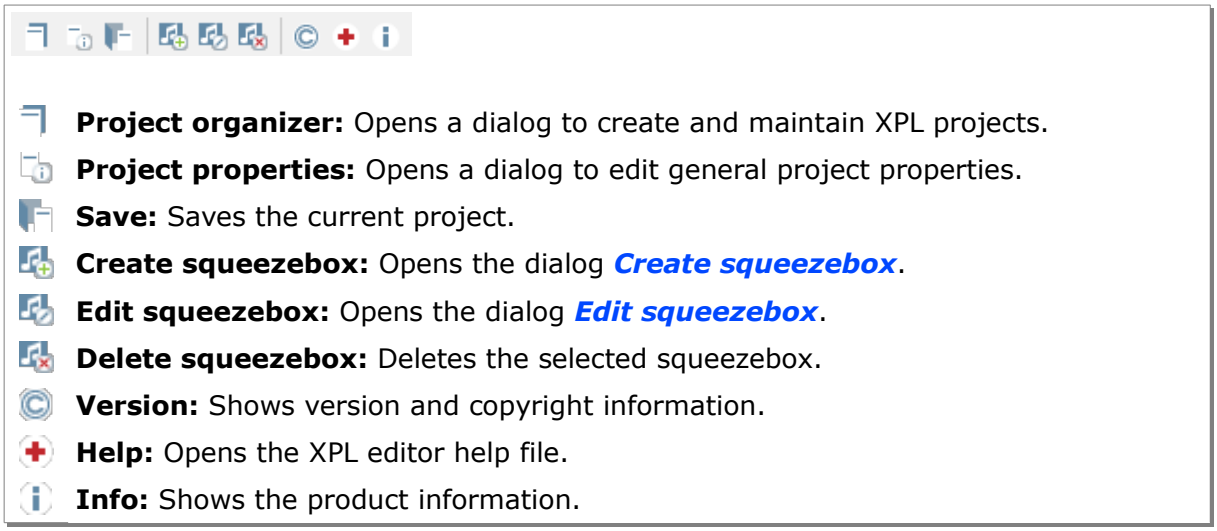


Help: Opens the XPL editor help file.

Info: Shows product and license information.

Version: Shows version and copyright information.

3.2 Toolbar



Project organizer: Opens a dialog to create and maintain XPL projects.

Project properties: Opens a dialog to edit general project properties.

Save: Saves the current project.

Create squeezebox: Opens the dialog ***Create squeezebox***.

Edit squeezebox: Opens the dialog ***Edit squeezebox***.

Delete squeezebox: Deletes the selected squeezebox.

Version: Shows version and copyright information.

Help: Opens the XPL editor help file.

Info: Shows the product information.

4 Functions

4.1 Project Organizer

In order to use the XPL editor, create first a project. A new project can be created in the ***Project organizer*** which starts automatically when the XPL editor is opened.

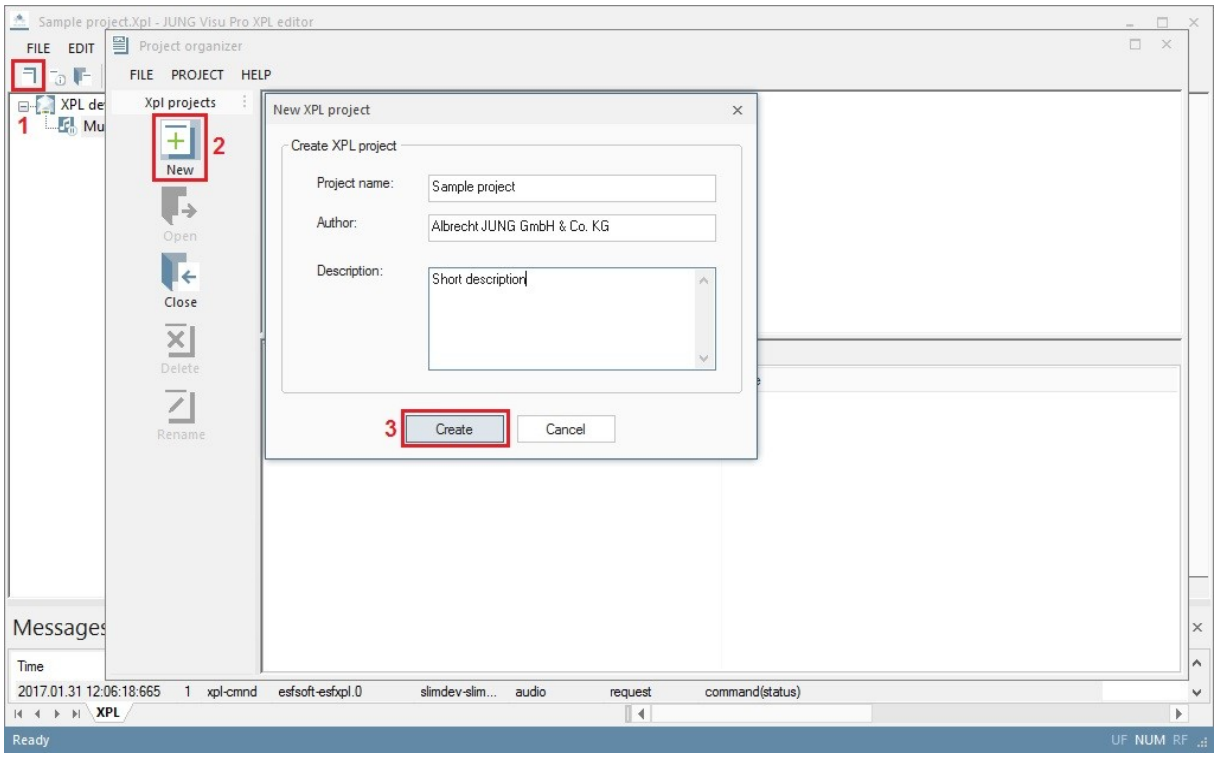
You can start the project organizer manually: choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar.

In the project organizer you can use the buttons on the sidebar or the entries under the menu item Project. The project organizer provides the following options:

- **New**: Create new XPL project.
- **Open**: Open the selected XPL project.
- **Close**: Close the currently opened project.
- **Delete**: Delete the selected project.
- **Rename**: Rename the selected project.

Create project

To create a new XPL project, click the button ***New*** in the sidebar or choose the menu item ***Project*** and click ***New***. The dialog ***New XPL project*** opens.



Enter the following information:

Project name: Unique name of the XPL project.

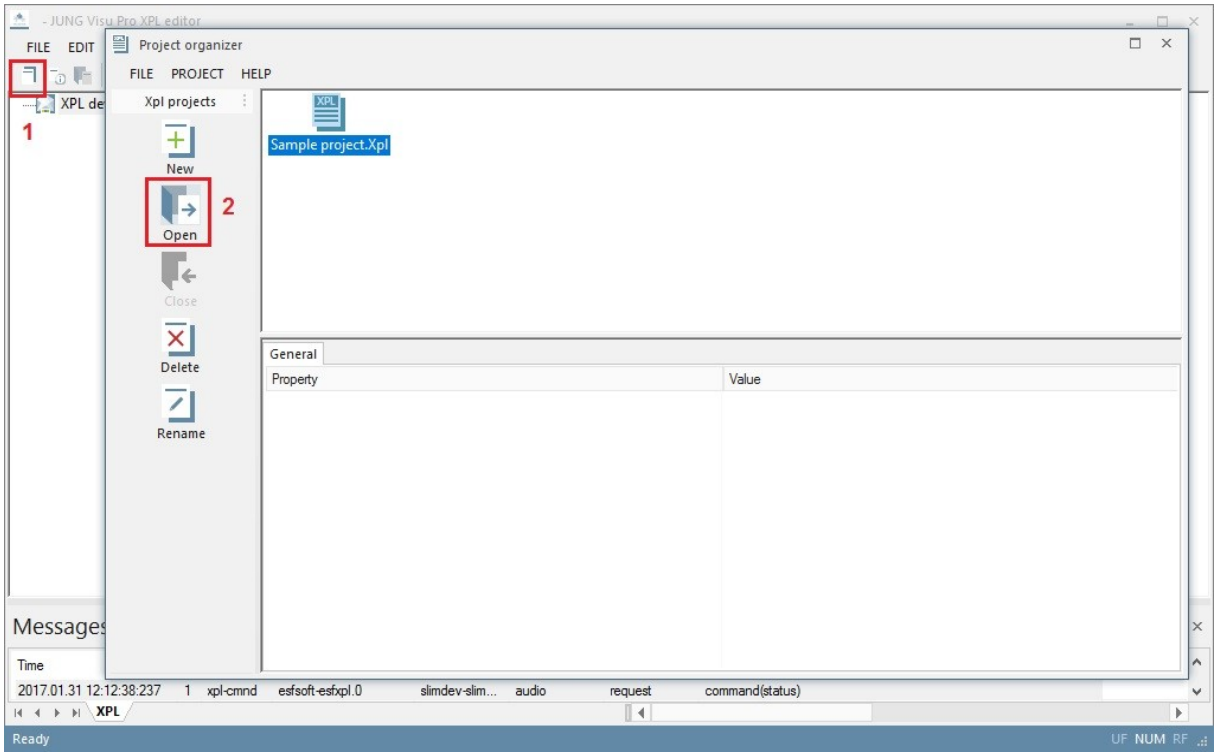
Author and description: The specification of the author and a description is useful but not mandatory. These entries can be entered or changed later.

Create: Press the button ***Create*** to complete the creation of the project. It is now available in the project list of the project organizer.

Open project

To load the newly created project into the XPL editor:

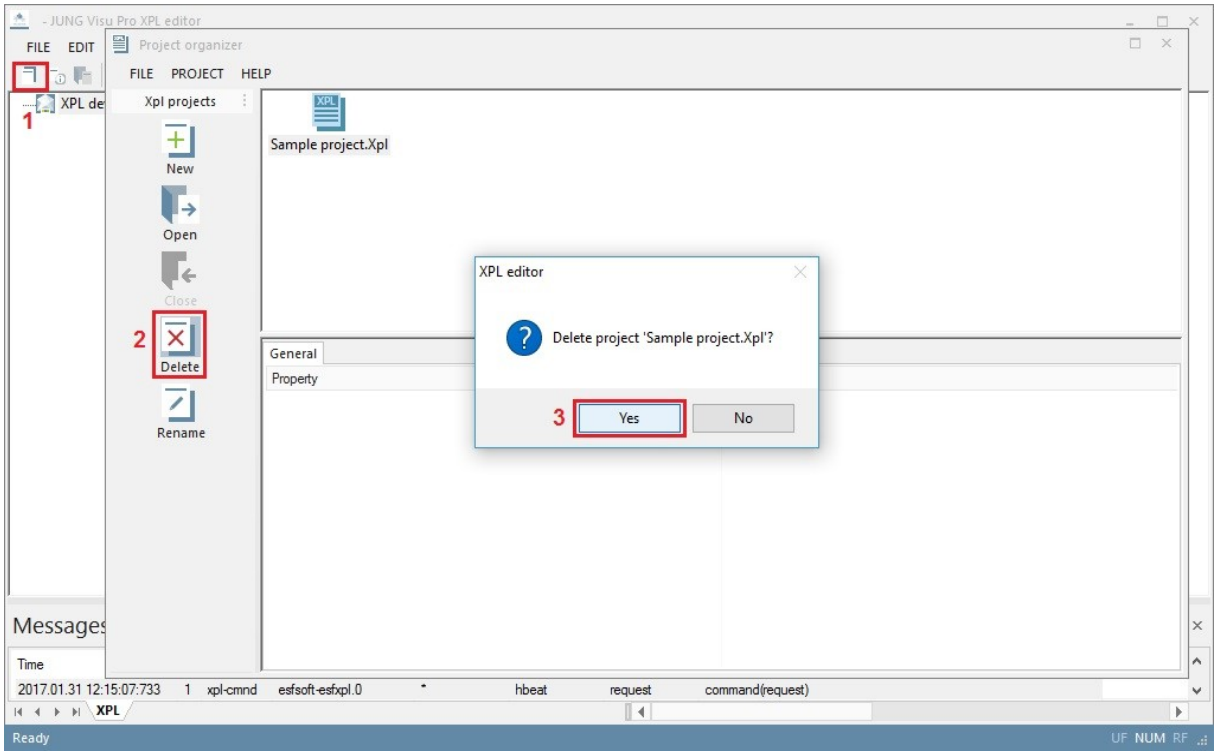
- double-click the project symbol or
- select the project and click the button ***Open*** in the sidebar or
- select the project, choose the menu item ***Project*** and click ***Open***.



The selected project is opened in the XPL editor.

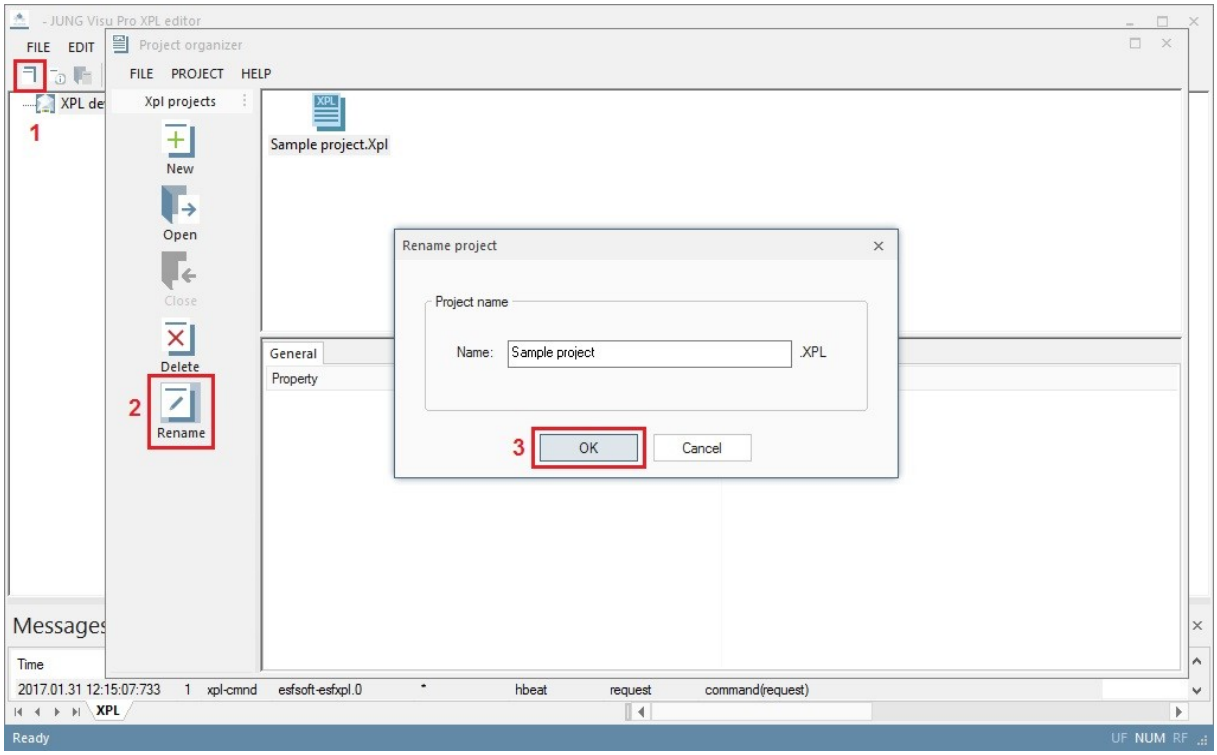
Delete project

To delete a project, select the project and click the button ***Delete*** in the sidebar or choose the menu item ***Project*** and click ***Delete***. Answer the confirmation prompt with the button ***Yes*** to complete the operation.



Rename project

To rename a project, select the project and click the button ***Rename*** in the sidebar or choose the menu item ***Project*** and click ***Rename***. The dialog ***Rename project*** opens.



Enter the new project name and click the button OK to submit the entry and to close the dialog. The extension ***.XPL*** is automatically appended to the project name.

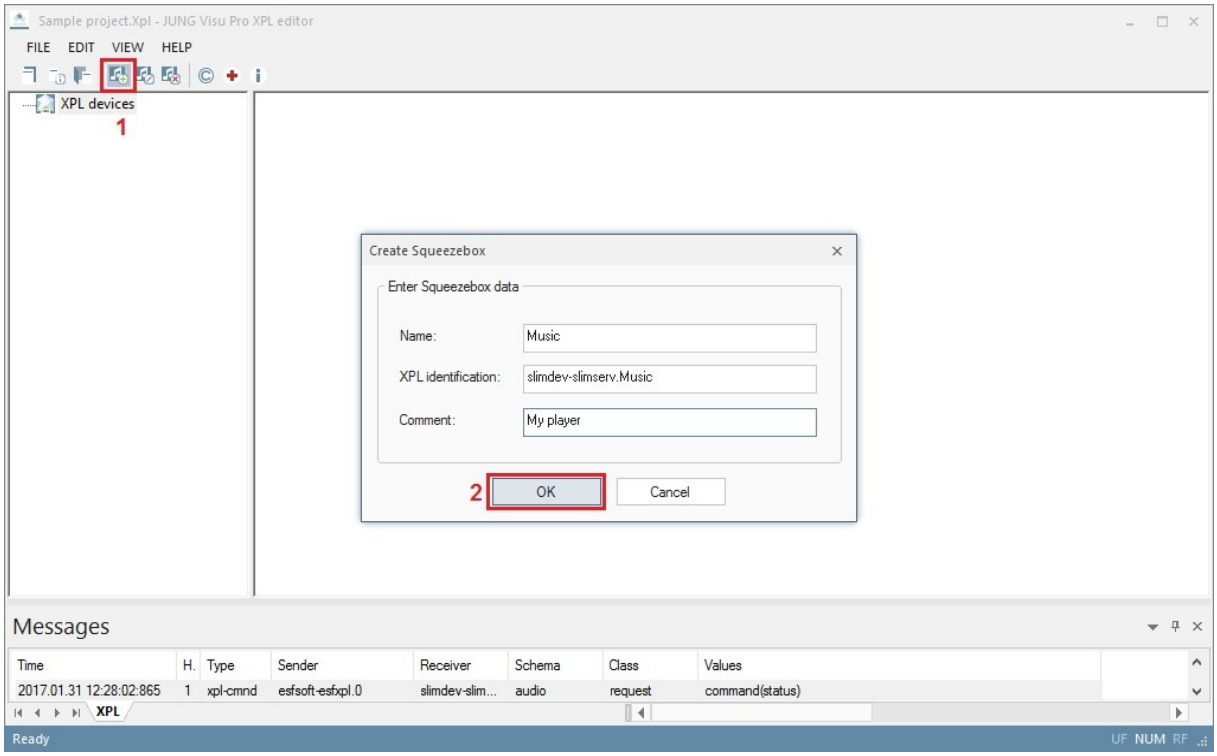
4.2 Squeezebox

In the XPL editor you can create, edit and delete Squeezebox devices.

Create Squeezebox

To create a Squeezebox, choose the menu item ***Edit*** and click ***Create Squeezebox*** or click the symbol in the toolbar.

The dialog ***Create Squeezebox*** opens.



Enter the following information:

Name: Name of the Squeezebox device in the editor.

XPL Identification: Identification of the device. Per default the first 8 characters of the device name are appended to 'slimdev-slimserv.' to form the complete XPL identification. Alternatively, the XPL Identification can be entered manually. 'slimdev-slimserv' as the first part specifies the Squeezebox Server, which provides the Squeezebox devices via the XPL protocol, the second part after the dot specifies the individual Squeezebox device.

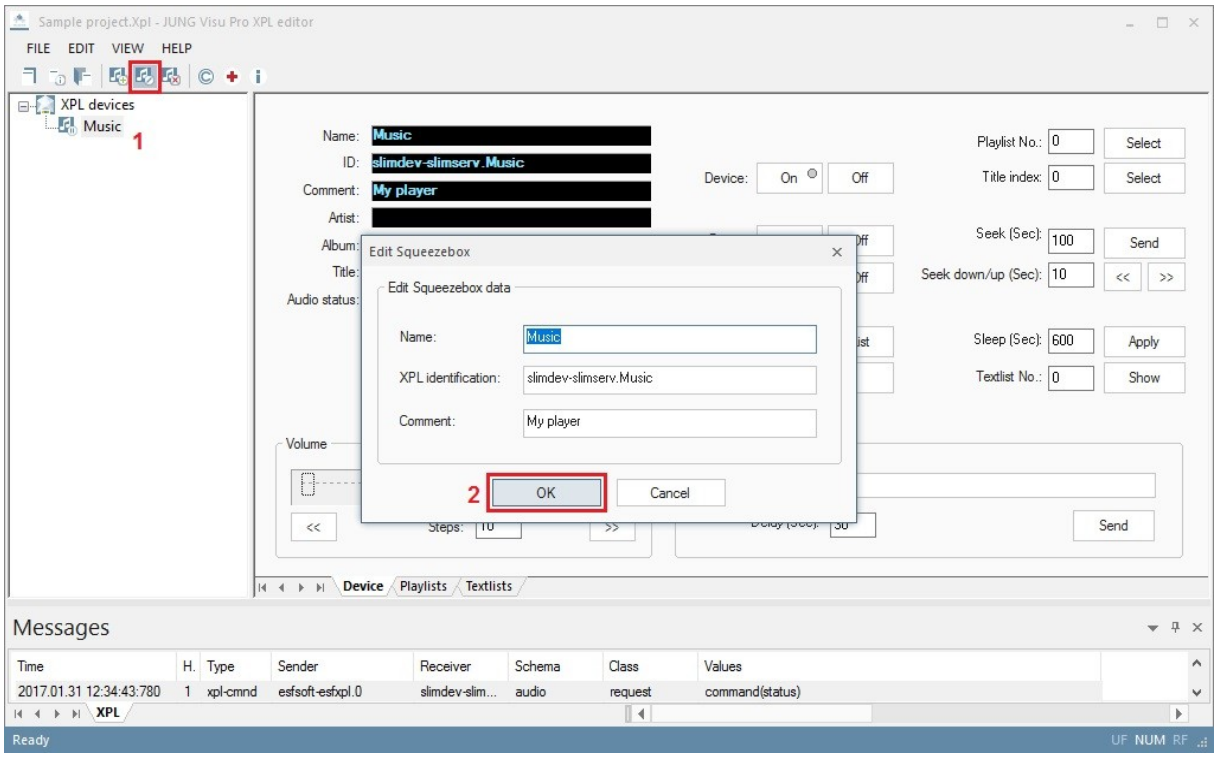
Note: *Make sure that the XPL identification matches the Squeezebox Server identity (first part, defined by the Squeezebox Server) and the name of the player in the Squeezebox Server (defined in the Squeezebox Server for the individual Squeezebox device).*

Comment: Optional comment for the Squeezebox device.

Click ***OK*** to submit the data and to complete the creation of the Squeezbox device in the editor.

Edit Squeezebox

To edit a Squeezebox, choose the menu item ***Edit*** and click ***Edit Squeezebox*** or click the button in the toolbar. The dialog ***Edit Squeezebox*** opens.



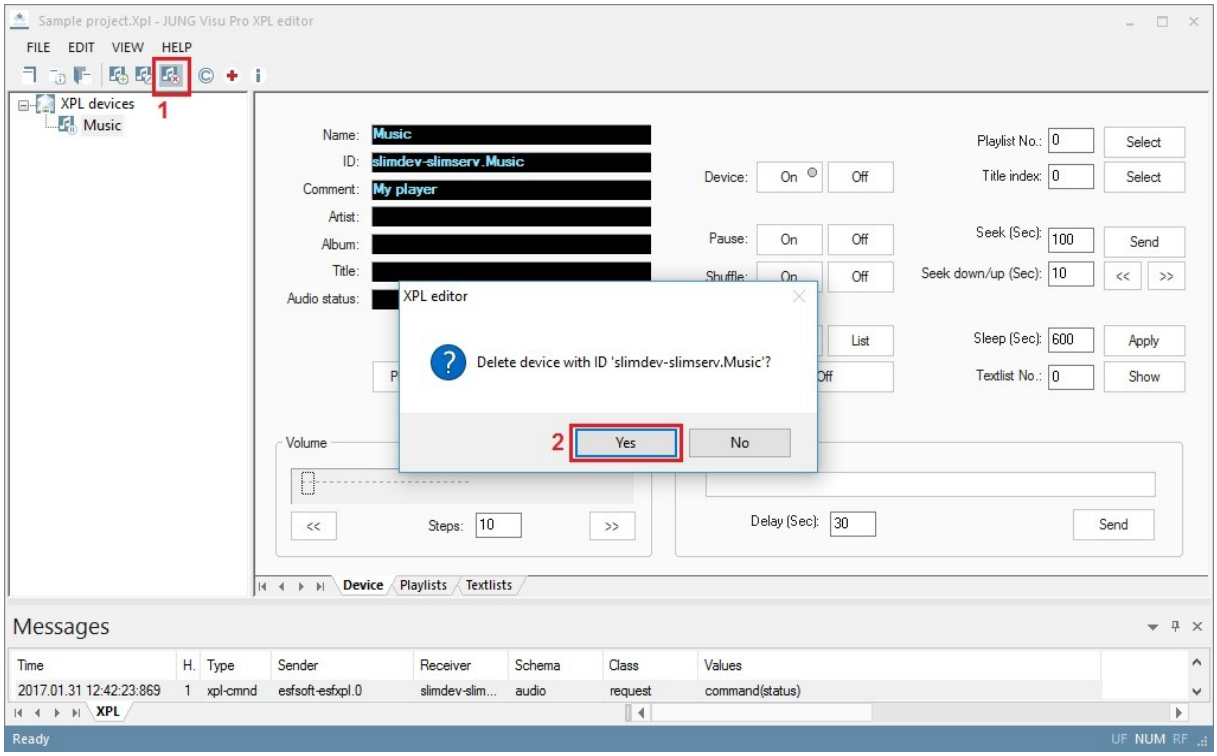
Edit the data of the Squeezebox and click OK to submit the changes and to close the dialog.

Note: *Make sure that the XPL identification matches the Squeezebox Server identity (first part, defined by the Squeezebox Server) and the name of the player in the Squeezebox Server (defined in the Squeezebox Server for the individual Squeezebox device).*

Delete Squeezebox

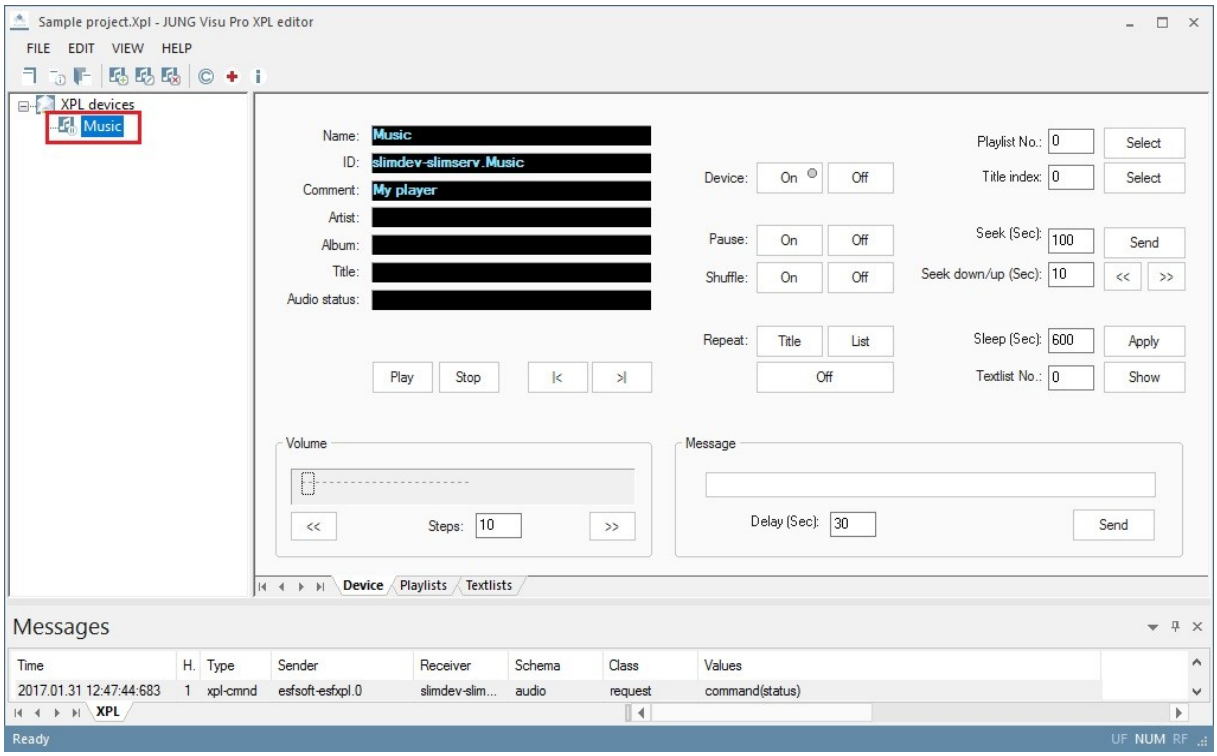
To delete a Squeezebox, select the device in the structure tree. Then choose the menu item Edit and click Delete Squeezebox or click the button in the toolbar.

Answer the confirmation prompt with the button ***Yes*** to complete the deletion operation.



4.3 Device Settings

To edit or view settings of a Squeezebox, select the device in the structure tree of the Squeezebox devices. The settings will be displayed in a window.



Play: Starts playing the current title in the selected play list.

Stop: Stops playing the current title.
|<: Jumps to the start position of the previous title in the current play list.

>|: Jumps to the start position of the next title in the current play list.

Volume: The slider control can be used to set the volume. Note that it is not updated automatically if the volume has been changed by other means.

Steps: Used to control volume up and down step by step.

Device: Turns device on or off.

Pause: Pause / continue playing.

Shuffle: Randomly select and play titles in the playlist.

Repeat: Repeat playing the selected title or play list until turned off.

Playlist No.: Select a play list by number. The number corresponds to positions in the list of playlists of the tab 'Playlists'. After entering the number, click ***Select*** to confirm the selection.

Title index: Select a title by number in the current playlist. After entering the number click ***Select*** to confirm the selection.

Seek (Sec): Continue playing the current title at the specified position, given as the number of seconds after the start of the title.

Seek down/up (Sec): Continue playing the current title a number of seconds before or after the current position.

Sleep: The device will be turned off after a number of seconds. Click ***Apply*** to start the timer.

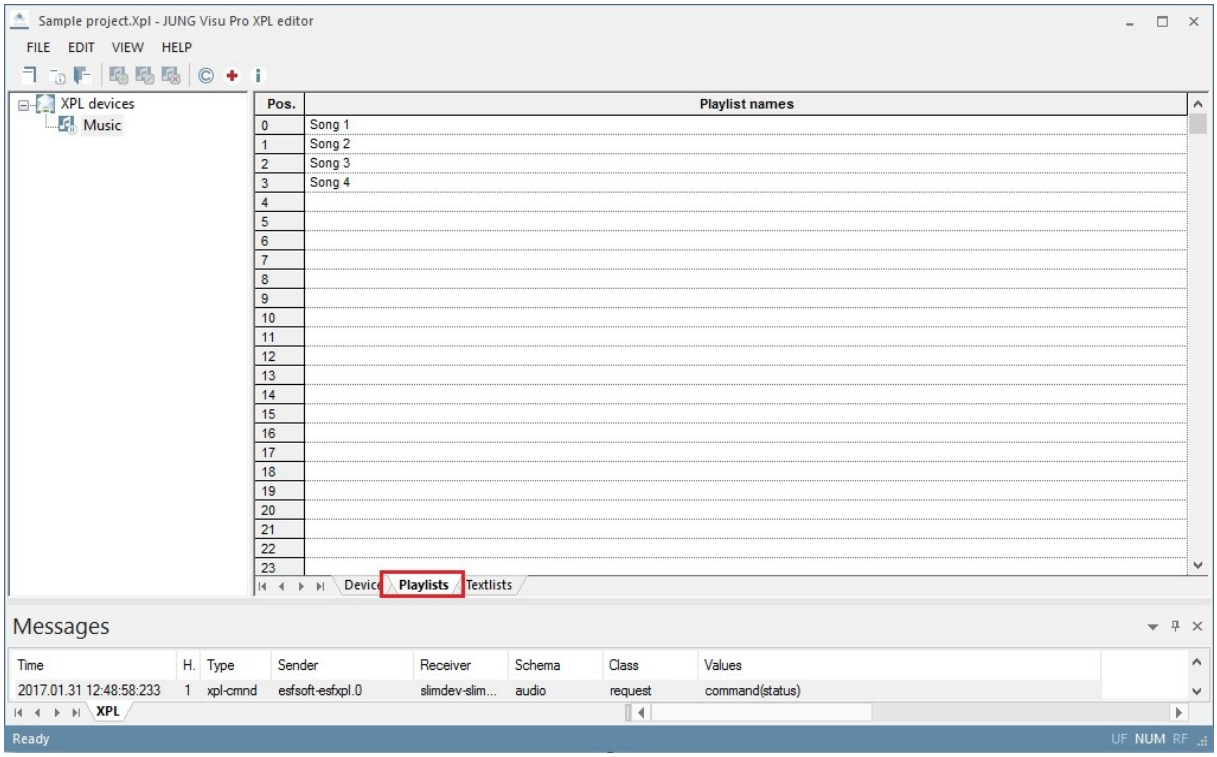
Textlist No.: Select a text by number. The number corresponds to positions in the list of texts of the tab ***Textlists.*** After entering the number and clicking ***Show***, the respective text will be displayed in the Squeezebox.

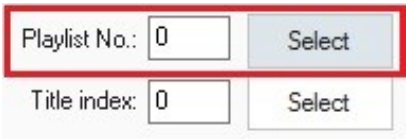
Message: Specify a text and a number of seconds. After clicking ***Send***, the message will be displayed in the Squeezebox device for the specified number of seconds.

4.4 Playlists and Textlists

Playlists

Playlists are collections of music defined in the Squeezebox Server software and may comprise individual music titles or complete albums. In the Squeezebox Server a name is assigned to each playlist. These names of playlists can be entered into the playlists window of the XPL editor. The position of a playlist in the list of playlists can be used to select a playlist by number, which can be the value of a parameter. (The use of a number for the selection of the playlist is more convenient for the process model than the use of the name of the playlist.)

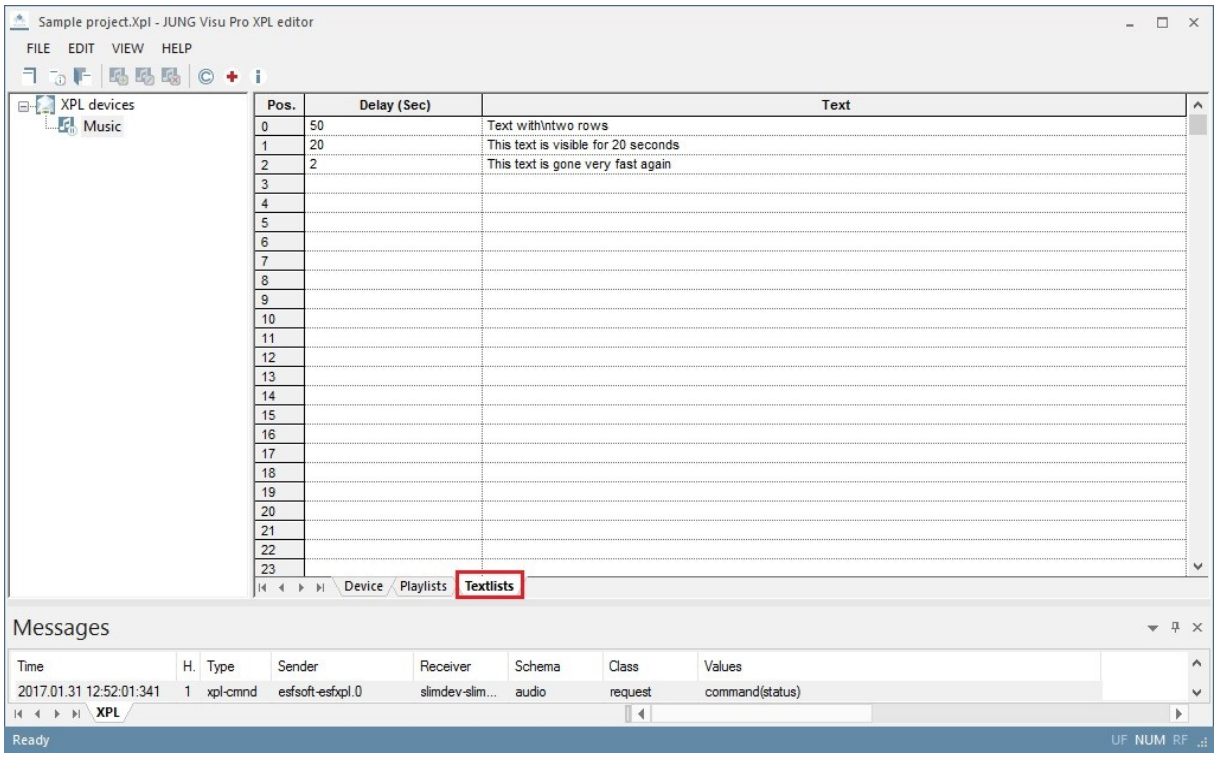




On the tab ***Device*** you can enter the **playlist number** you want to be played.

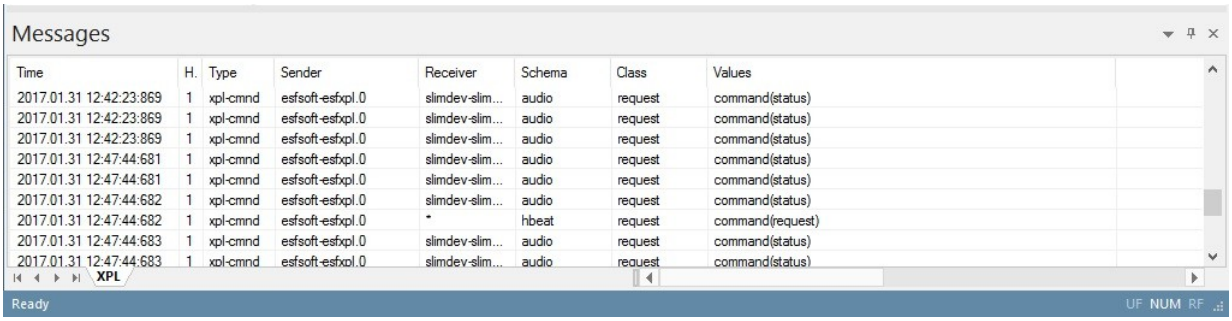
Textlists

The textlists window shows a table with each row comprising a time period in seconds and a text. On the tab ***Device*** a text and the corresponding time can be selected by the row position in the textlists tab to show the selected text for the number of seconds at the Squeezebox device. The text can be displayed in two rows. To separate two rows, use shift+enter.



4.5 Message List

To show or hide the message list of the XPL editor, choose the menu item ***View*** and click ***Messages*** . The message list displays sent and received telegrams.



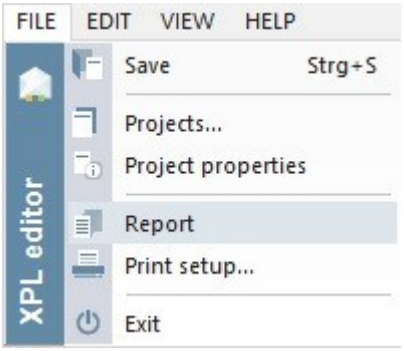
To delete the content of the message list, choose the menu item ***View*** and click ***Delete Messages***.

To enable or disable the auto scroll function, choose the menu item ***View*** and click ***Show last message***. When turned on, the window will always automatically scroll to the last message.

5 Reports

- Reports provide overviews of the project.
- Reports may be printed (with print preview).

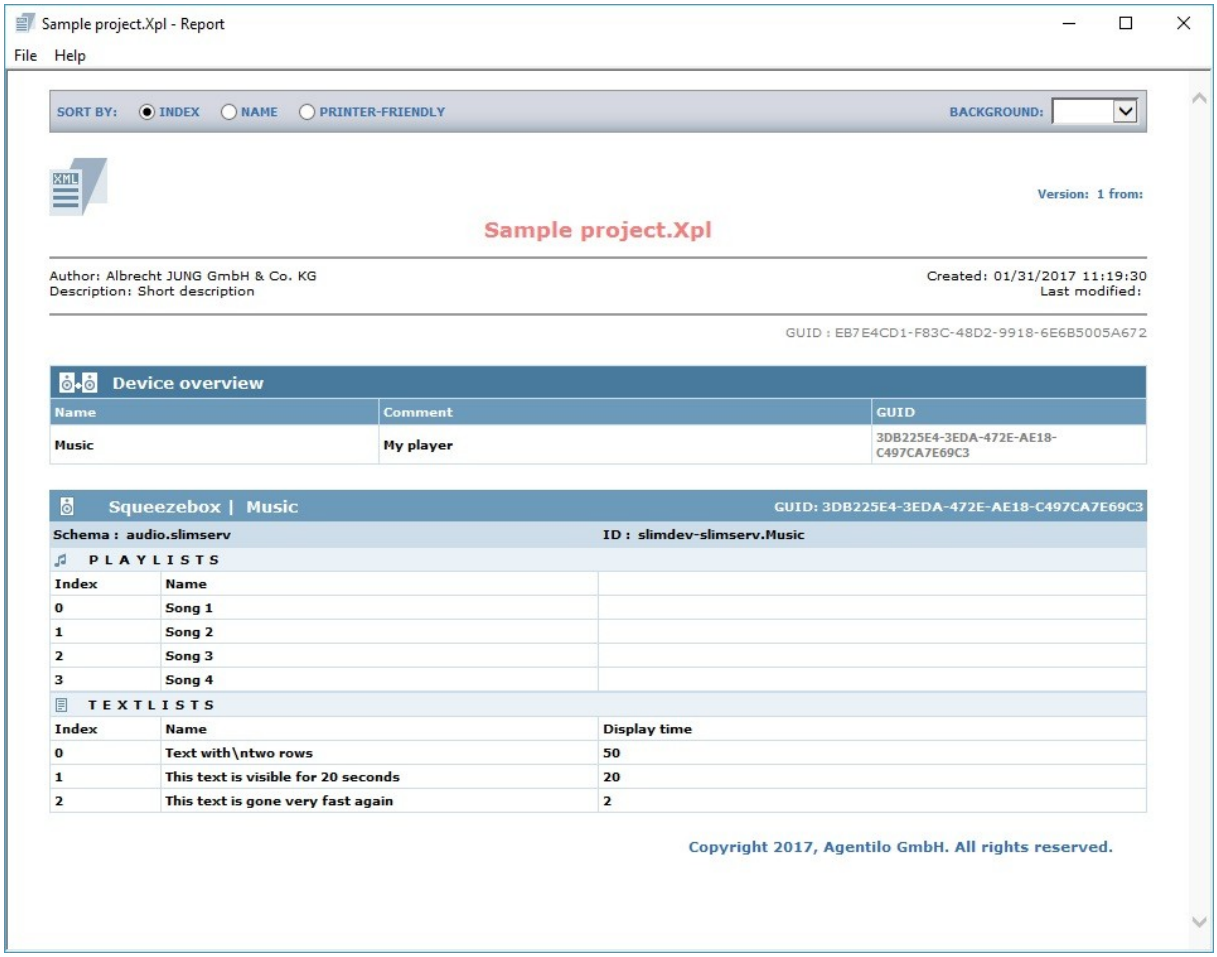
5.1 Create Report



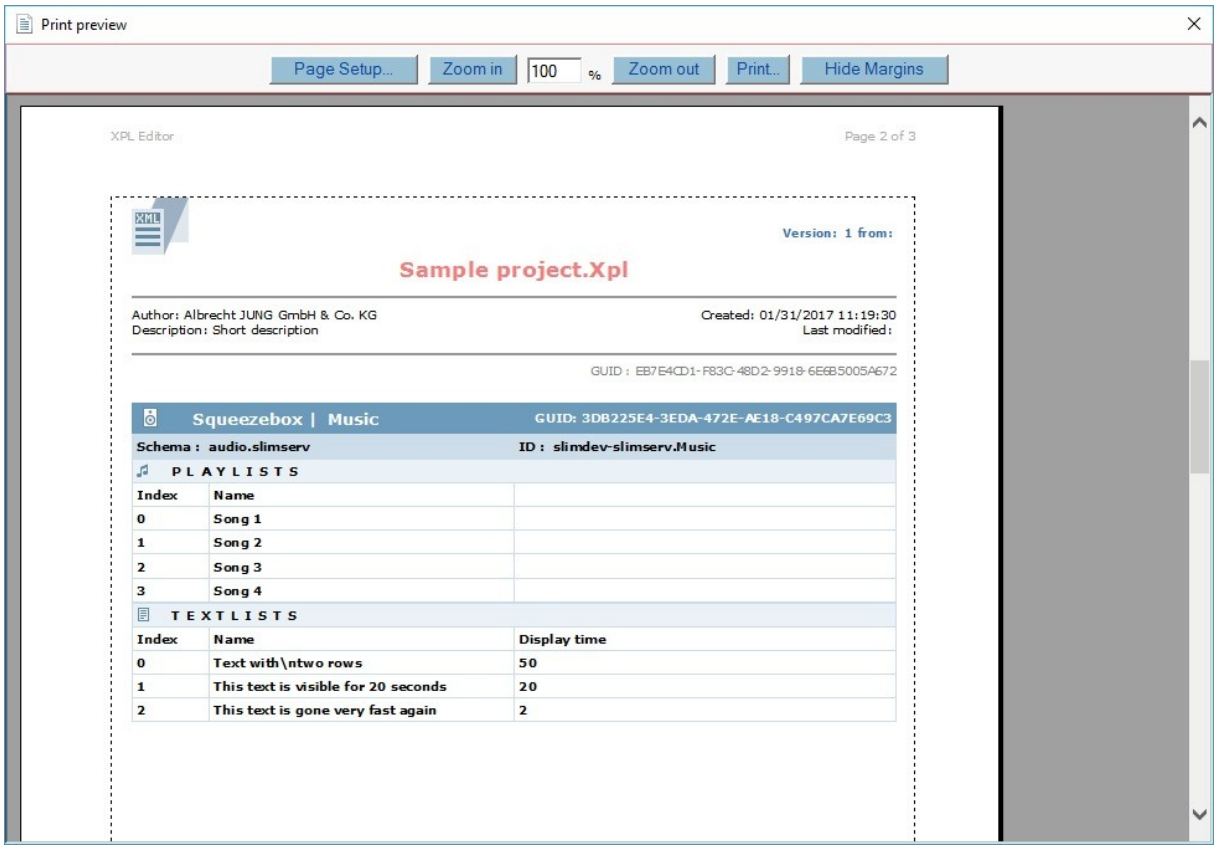
To create a report for the current project, choose the menu item ***File*** and click ***Report***.

The menu at the top offers different options:

- **Index:** Shows the devices, playlists and textlists sorted by Index.
- **Name:** Shows the devices, playlists and textlists sorted by name.
- **Printer friendly:** Black and white display of the devices, playlists and textlists.


Sample project.Xpl - Report
File Help
SORT BY: INDEX NAME PRINTER-FRIENDLY
BACKGROUND:
Version: 1 from:
Sample project.Xpl
Author: Albrecht JUNG GmbH & Co. KG
Description: Short description
Created: 01/31/2017 11:19:30
Last modified:
GUID : EB7E4CD1-F83C-48D2-9918-6E6B5005A672
Device overview
Name
Comment
GUID
Music
My player
3DB225E4-3EDA-472E-AE18-C497CA7E69C3
Squeezebox | Music
GUID: 3DB225E4-3EDA-472E-AE18-C497CA7E69C3
Schema : audio.slimserv
ID : slimdev-slimserv.Music
PLAYLISTS
Index
Name
0
Song 1
1
Song 2
2
Song 3
3
Song 4
TEXTLISTS
Index
Name
Display time
0
Text with\ntwo rows
50
1
This text is visible for 20 seconds
20
2
This text is gone very fast again
2
Copyright 2017, Agentilo GmbH. All rights reserved.

5.2 Print Preview



The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

With these , margin markers the user has visual control over the margin settings.

JUNG Visu Pro
Process Model Editor

Content:

1 TECHNICAL REQUIREMENTS....3

1.1 Hardware....3
1.2 Operation Systems....3
1.3 Process Interface....3
1.4 Import from ETS....3
1.5 Internet Explorer, DirectX and .Net Framework....3

2 INTRODUCTION....4

3 USER INTERFACE....5

3.1 Menu....5
3.2 Toolbar....9
3.3 Tool Windows....10
3.3.1 Tasks Tool Window....10
3.3.2 Catalog Tool Window....11

4 CREATE LOGICAL PROJECT....14

4.1 Project Organizer....14
4.2 Project Properties....17
4.3 Import from Process Interface Projects....18
4.3.1 Folder Bus State....19
4.4 Conversion of a KNX Project 1.0 to 2.0....20
4.5 Save the Process Model....21
4.6 Update from Process Interface Projects....22
4.7 Properties of Process Variables....23
4.8 Calculated Process Variables....29

5 ALARM LIMITS....31

6 EMAIL NOTIFICATIONS....33

6.1 Create Email Notification....33
6.2 Delete Email Notification....35
6.3 General Email Settings....35
6.4 Extended Email Settings....36
6.5 Checking Formulas and Conditions....38

7 ARCHIVES....40

7.1 Event Archives....40
7.2 Interval Archives....42
7.3 Consumption Data Archives....44
7.4 View Archives....46
7.5 Export to CSV File....47
7.6 Configuration of CSV Export....48
7.7 Graph Export....49

8 FUNCTIONS....50

8.1 Starting Behavior....51
8.2 Add Function....51
8.3 Logical Gate....53
8.4 Mathematics....55
8.5 Sequence....57
8.6 Status Object....60
8.7 Scene....62
8.8 Scene with Memory....63
8.9 Scene with Memory and Status....66
8.10 Forwarding....68
8.11 Forwarding Collection....70
8.12 If Then....71
8.13 16 Channel Weekly Time Switch....73
8.14 Counter....75
8.15 Timing Relay....77
8.16 Automatic Guard....79
8.17 Own Consumption Optimisation....81
8.18 Voice Service Group....85
8.18.1 Interpretation of the Values....86

9 SET UP VOICE SERVICE....87

9.1 Settings for the Voice Service....87
9.2 Configure the Process Variable for Voice Control....87

10 REPORTS....88

10.1 Create Report....88
10.2 Print Preview....91
10.3 Voice Service Report....92

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany
Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.
Interfaces	Interface for the connection to the process

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Process Interface

The connection to the KNX bus requires the installation of a suitable version of the FALCON driver of the KNX Association and the appropriate interfaces.

Optionally, the visualization system is equipped with an OPC (OLE for Process Control) client, so that instead of or additionally to KNX, OPC servers – which are available for a multitude of automation systems – can be used for the connection to the process.

1.4 Import from ETS

ETS5	YES, use of ETS, project export
ETS4	YES, use of ETS, project export
ETS3, ETS2 version 1.3	YES, use of ETS, OPC export
ETS2 versions 1.1, 1.2 and 1.3	YES, use of print reports redirected to file
Earlier ETS versions	NO

1.5 Internet Explorer, DirectX and .Net Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 Introduction

A process model describes technical processes from an application point of view.
Whereas the process interfaces implement access to connected devices, the logical process model describes the desired functionality based on parameters given by these devices. At this level, additional parameters may be defined and automatic control functions may be set up to supplement the behavior of the devices.

Whereas the process interface provides for instance access to a room's temperature and heating control, the logical process model defines archives of the temperatures measured over a time period and relates the desired room temperature to the occupancy state – perhaps planned in a calendar – by setting the control value for the heating device.

Several process interfaces may be combined in one logical process model.

The process model is also the level to define alarm and warning ranges for parameters and to set up the alarm behavior which may comprise sending notifications via email.

After the installation of the visualization system a program group is created in the Start Menu. The default settings can be adopted or manually changed during installation. You can start the control panel of the visualization system from the WINDOWS program manager. To start an application, select the list entry ***Function Modules*** in the sidebar of the control panel and double-click the icon of the application you wish to start.

In order to clearly distinguish between the different applications, different colors were used. The icon of the **process model editor** contains a green colored bar .

The simplest process model:
The simplest process model only imports a process interface project, i.e. a KNX or OPC project, and supplements no additional functions.
In this case, the process model should be created, connected to a process interface project and saved. Then it can already be used by other applications relying on a process model: currently the visualization editor and the calendar program. More functions can be added later to the process model.

Proceeding:
The process interface (i.e. the KNX editor) provides the raw data, that now are used for different purposes: Some values are archived, others are monitored and should optionally trigger emails. Other types of values are used in scenes or sequences, are the object of evaluation or serve the visualization of a building or a process and thereby enable the user to communicate with the process.
Therefore it is necessary to extend the KNX group addresses by archives, logical functions, calculation formulas or email notifications. The process model editor allows the user to set these functions. The process model that results from the extension of the group addresses constitutes the basis for the visualization.

The creation of a process model follows the following 4 steps:

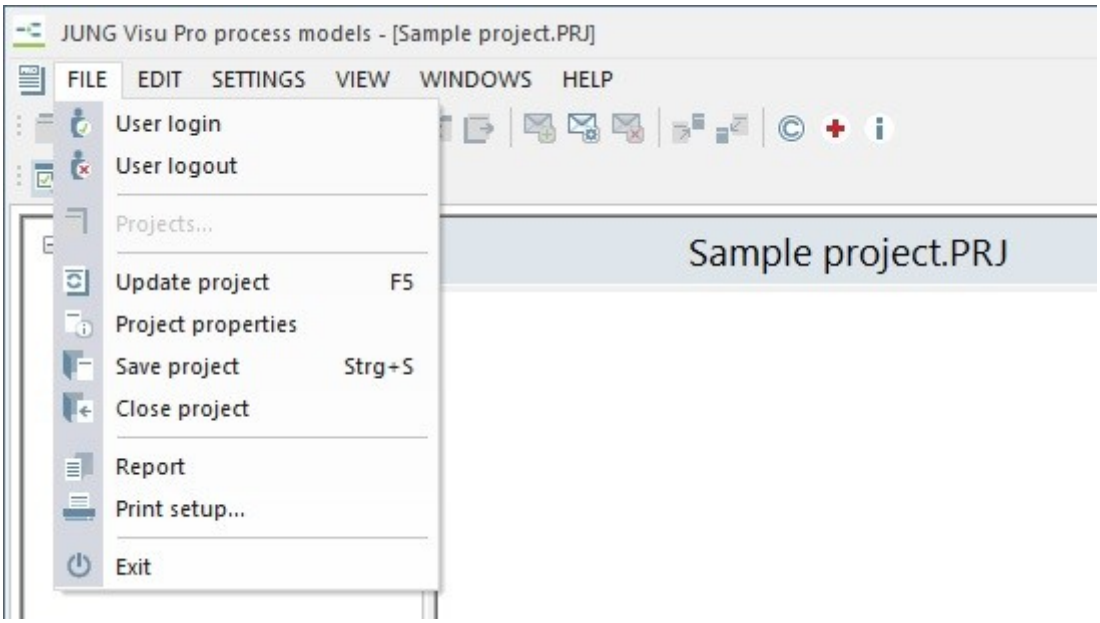
1. Create a process model project.
2. Connect the associated KNX project and additional functions with the process model.
3. Create archives, calculation formulas or email notifications.
4. Save the project.

Step 3 can be omitted if neither formulas nor email notifications shall be generated. An existing process model can be modified in the process model editor at any time. For further information on the individual steps, see the following sections.

3 User Interface

3.1 Menu

Menu item *File*



User login: Opens a dialog to log in a user to the visualization system.

User logout: Logs current user out of the system.

Projects: Opens the project management dialog. In this dialog projects can be created and maintained.

Update project: Updates data imported from the logical process model. Shortcut F5.

Project properties: Opens a dialog to define or update general project properties.

Save project: Saves the current project.

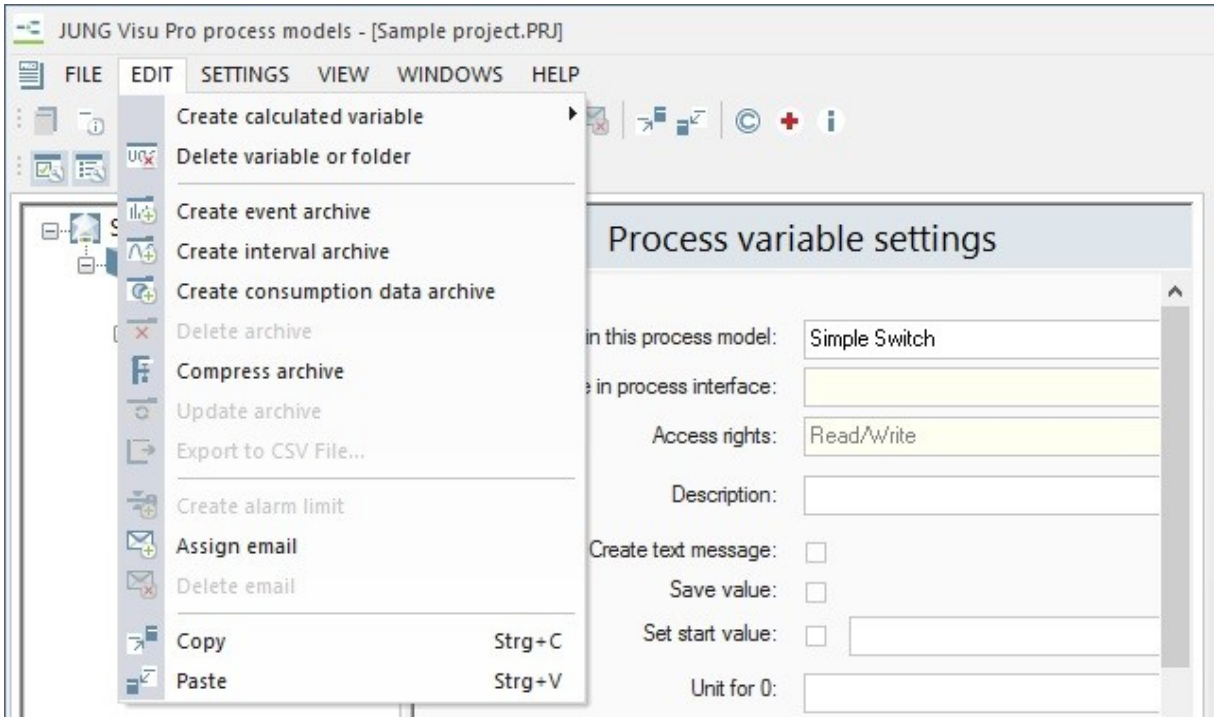
Close project: Closes the process model project.

Report: Generates a report for the project.

Print setup: Selection of printer and setting of printer properties.

Exit: Quits the process model editor. Shortcut ALT+F4.

Menu item *Edit*



Create calculated variable: Create a calculated process variable. The process variable may be a binary (shortcut CTRL + B), an analog (shortcut CTRL + A) or a text string (shortcut CTRL + T) variable.

Delete variable or folder: Deletes the selected process variable or selected folder of the process variables.

Create event or interval archive: Creates a new event or interval archive for the selected process variable.

Create consumption data archive: Creates a new consumption data archive.

Delete archive: Deletes the selected archive.

Compress archive: Compresses the archive database of the current project.

Update archive: Updates the display of the selected archive.

Export to CSV file: Exports the data in the archive data viewer to a CSV file.

Create alarm limit: Creates an alarm limit for the selected analog variable.

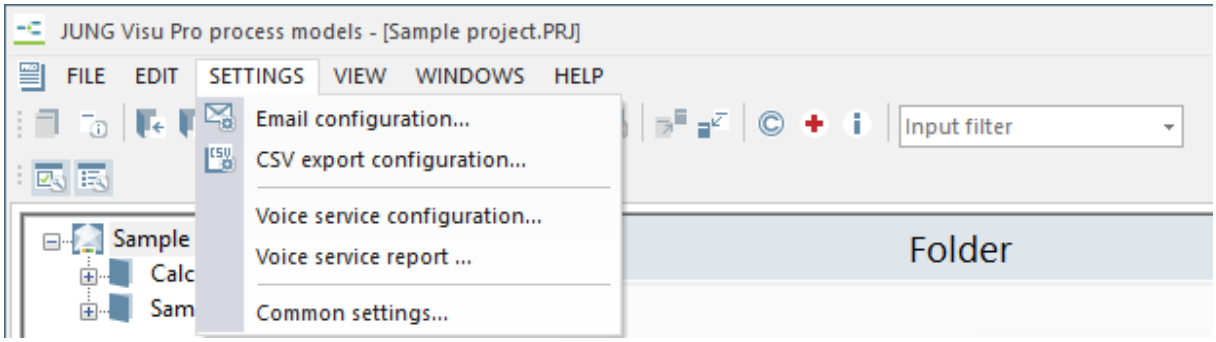
Assign email: Creates an email notification for the selected process variable. The email will be sent when the process variable attains a certain value predefined by the user.

Delete email: Deletes email notification.

Copy: Copies selected items to clipboard.

Paste: Pastes items from clipboard.

Menu item *Settings*



Email configuration: Opens the dialog ***email settings*** for the configuration of the SMTP server and the name and address of the sender.

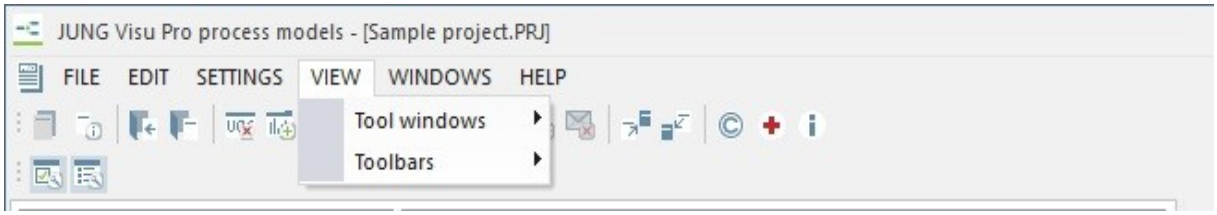
CSV export cofiguration: Opens the dialog to edit the CSV export options.

Voice service configuration: Opens the dialog ***Voice Service Settings*** for the configuration of the connection to a voice assistant.

Voice service report: Creates a voice assistant datapoint report as a PDF file at the chosen path.

Common Settings: Contains common settings for creating process variables, such as e.g. the setting of "save value" when creating calculated process variables.

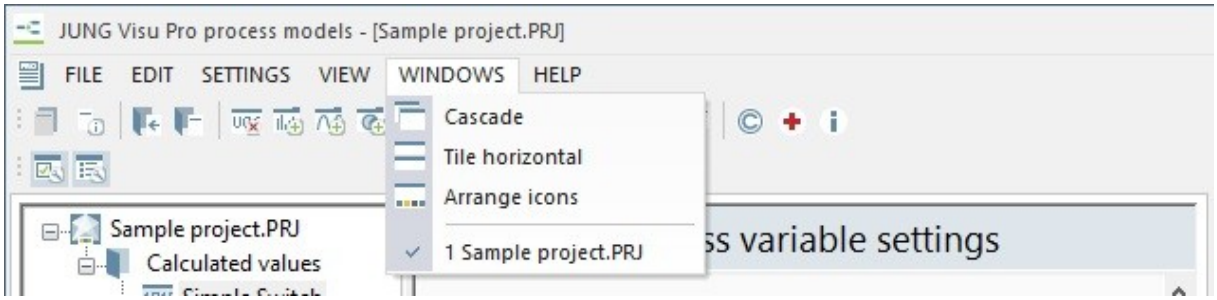
Menu item *View*



Tool windows: Toggles the visibility of the different tool windows.

Toolbars: Toggles the visibility of the different toolbars.

Menu item *Windows*

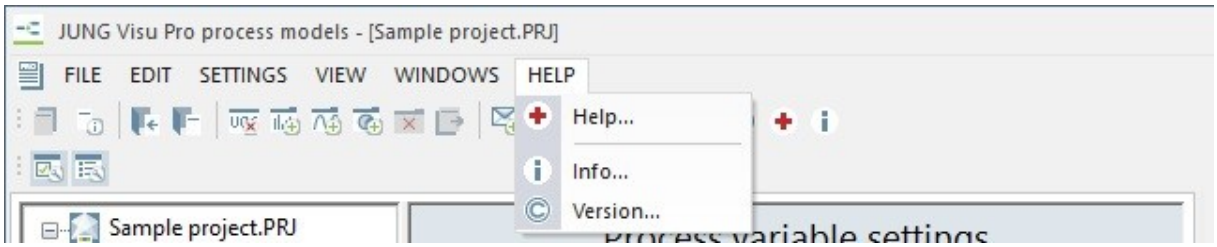


Cascade: The worksheet windows overlap.

Tile horizontal: The worksheet windows are tiled horizontally.

Arrange icons: Arranges the icons of minimized windows.

Menu item *Help*

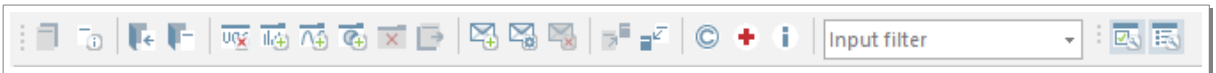


Help: Opens the help file.

Info: Shows product and license information.

Version: Shows version and copyright information.

3.2 Toolbar



Project organizer: Opens the project organizer dialog. In this dialog projects can be created and maintained.

Project Properties: Opens a dialog to define or update general project properties.

Close project: Closes the process model project.

Save project: Saves current project.

Delete variable or folder:
Deletes the selected process variable or the selected folder of process variables.

Create event archive:
Creates a new event archive for the selected process variable.

Create interval archive:
Creates a new interval archive for the selected process variable.

Create consumption data archive:
Creates a new consumption data archive for the selected process variable.

Delete archive: Deletes the selected archive.

Export to CSV file:
Exports the data in the archive data viewer to a CSV file.

Create email: Creates an email notification for the selected process variable. The email will be sent when the process variable attains a certain value predefined by the user.

email configuration: Opens the dialog ***email settings*** for the configuration of the SMTP server and the name and address of the sender.

Delete email: Deletes the email notification.

Copy: Copy selected items to clipboard.

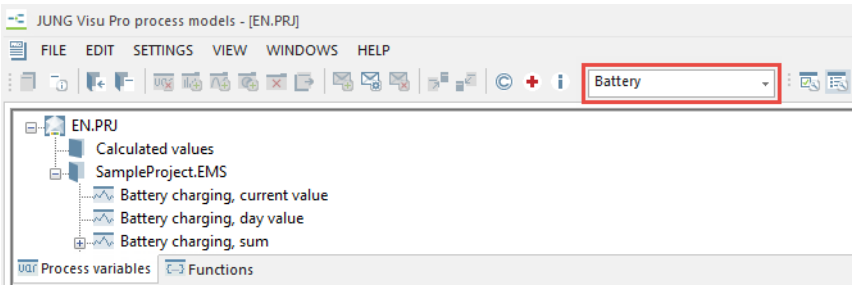
Paste: Paste items from clipboard.

Version: Shows version and copyright information.

Help: Opens the help file.

Info: Shows the product information.

Enter filter: To filter the process variables or functions, click into the search box ***Enter Filter*** and enter a search term. Then press the ***Enter*** key.



Tasks tool window: Toggles the visibility of the tasks tool window.

Catalog tool window: Toggles the visibility of the catalog tool window.

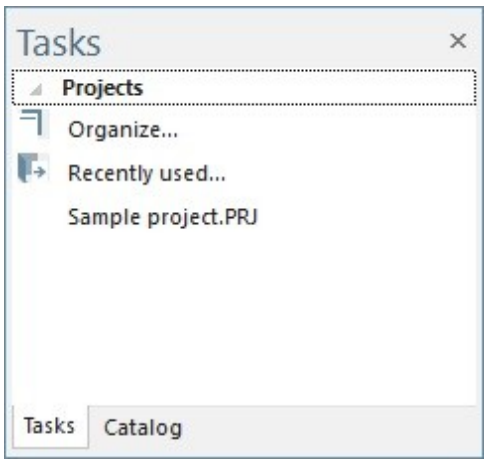
3.3 Tool Windows

The editor comprises tool windows which provide overviews, assistance and easy access to frequently performed actions. According to the current situation, the contents of the tool windows change automatically.

Choose the menu item ***View*** and click ***Tool windows*** to show or hide the individual tool windows.

Alternatively, the visibility of the individual tool windows can be toggled by clicking on the respective buttons in the toolbar. Choose the menu item ***View*** and click ***Toolbars*** to hide or show the toll windows toolbar.

3.3.1 Tasks Tool Window



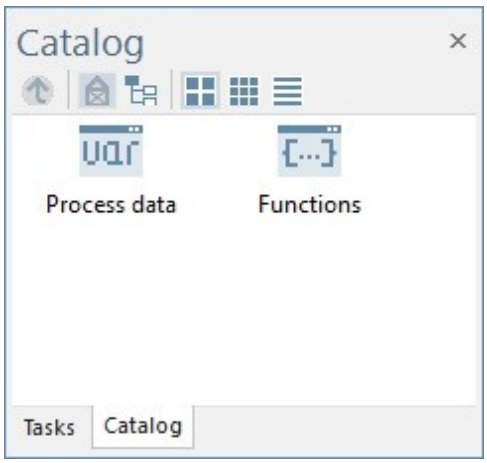
Tasks:

The tasks tool window shows frequently performed tasks which can be selected with a single click. The contents of this window depend on the current state.

In case no project has been loaded yet, open the dialog ***Project organizer***. Select a project from the recently used projects.

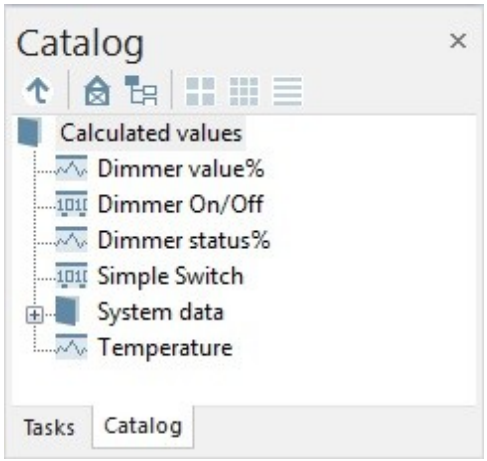
In case a project has been opened, the option ***Close*** can be selected.

3.3.2 Catalog Tool Window



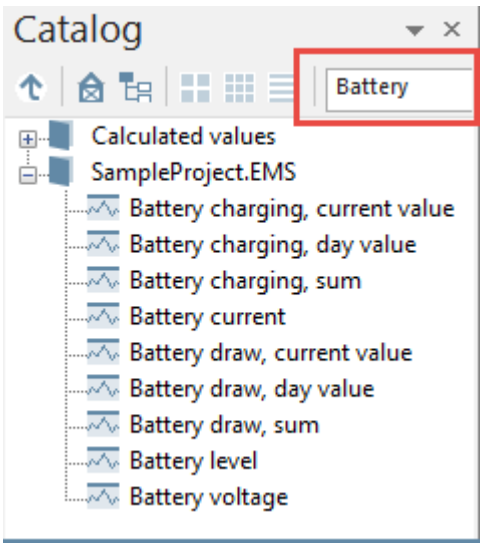
Catalog:

The catalog tool window provides an overview of the process data and the configurable functions of the process model.



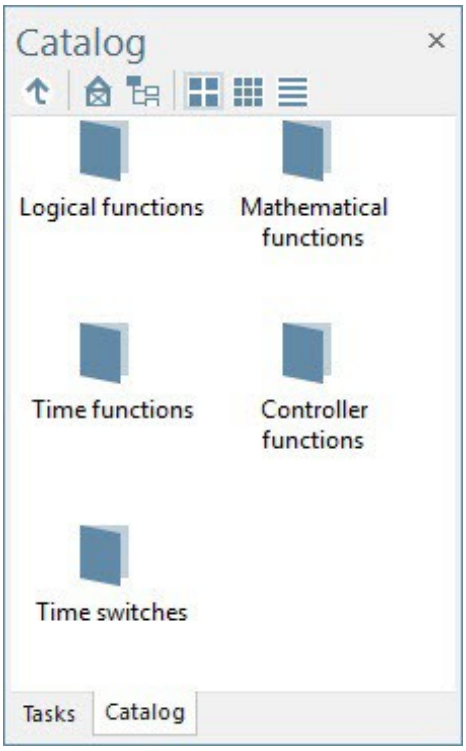
Process interfaces and data points:

The process variables can be dragged to the configuration windows of the process model functions.



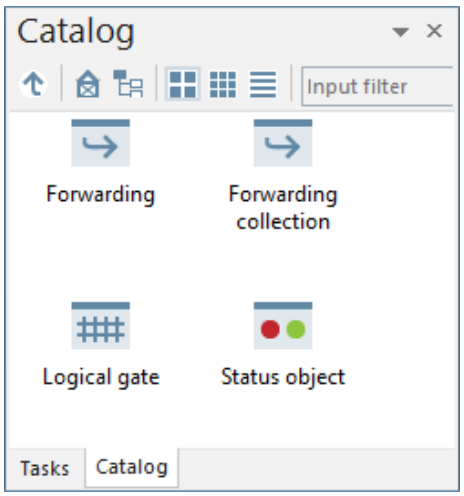
Note:

To filter the process variables, click into the search box ***Enter Filter*** and enter a search term. Then press the ***Enter*** key.



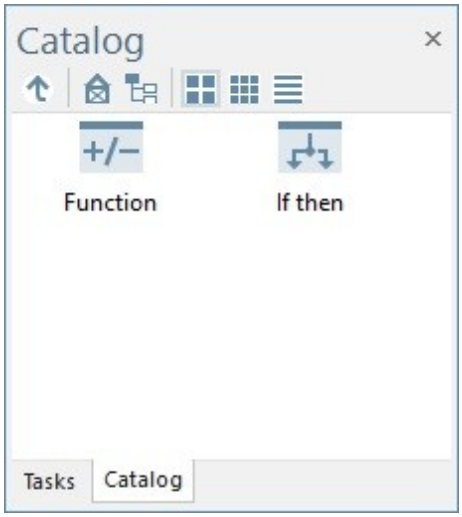
Functions:

Overview of different categories of functions which can be created in the process model.



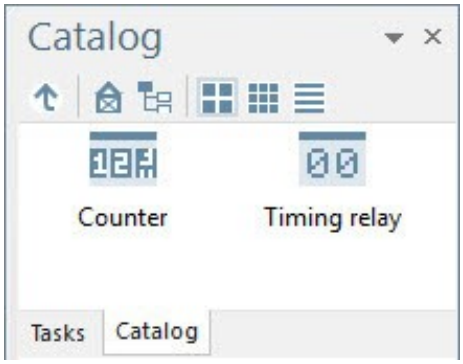
Logical functions:

In order to add a function to the function overview, drag the function to the main window of the process model editor.



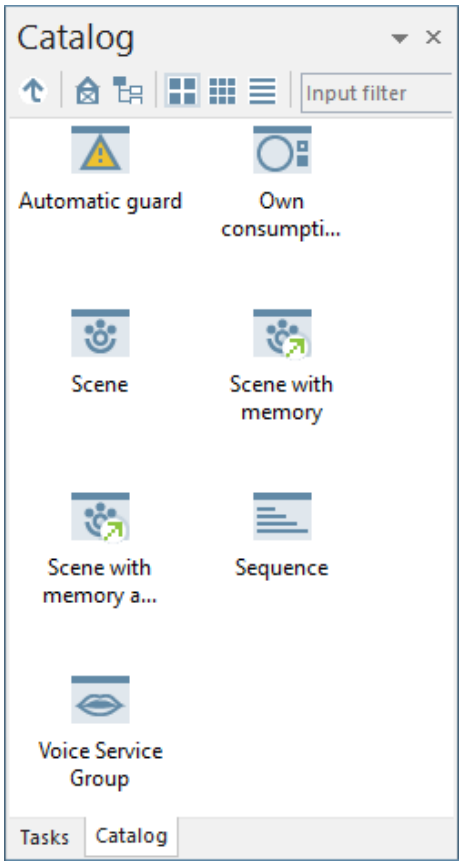
Mathematical functions:

In order to add a function to the function overview, drag the function to the main window of the process model editor.



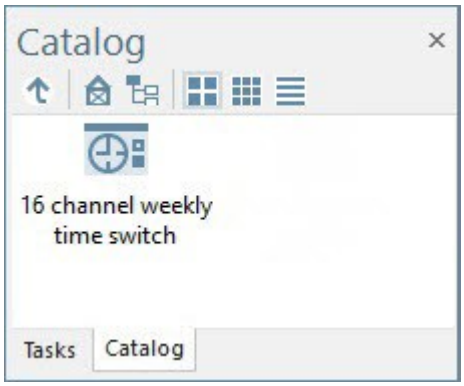
Time functions:

In order to add a function to the function overview, drag the function to the main window of the process model editor.



Controller functions:

In order to add a function to the function overview, drag the function to the main window of the process model editor.



Time switches:

In order to add a function to the function overview, drag the function to the main window of the process model editor.

4 Create Logical Project

4.1 Project Organizer

Choose the menu item ***File*** and click ***Projects*** or click the button in the toolbar or in the tasks tool window. The dialog ***Project organizer*** opens. In this dialog you may create, open, delete or rename process model projects.

In the project organizer you can use the buttons in the sidebar or the entries under the menu item ***Project***. The project organizer provides the following options:

- **New:** Create new process model.
- **Open:** Load an existing project into the process model editor.
- **Close:** Close the currently open project.
- **Delete:** Delete an existing project.
- **Rename:** Rename an existing project.

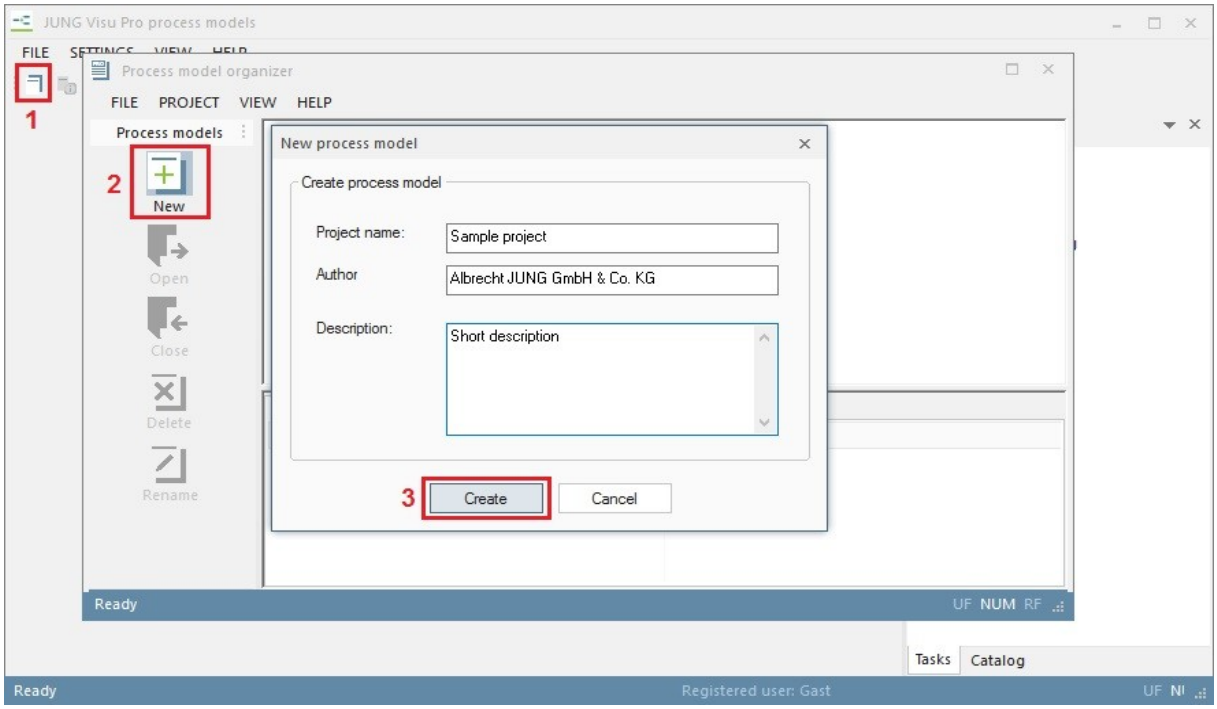
Create process model:

Click the button ***New*** in the sidebar. The dialog ***New process model*** opens.

Enter the data of the new process model:

Project name: Enter a unique name for the process model project. This specification is mandatory.

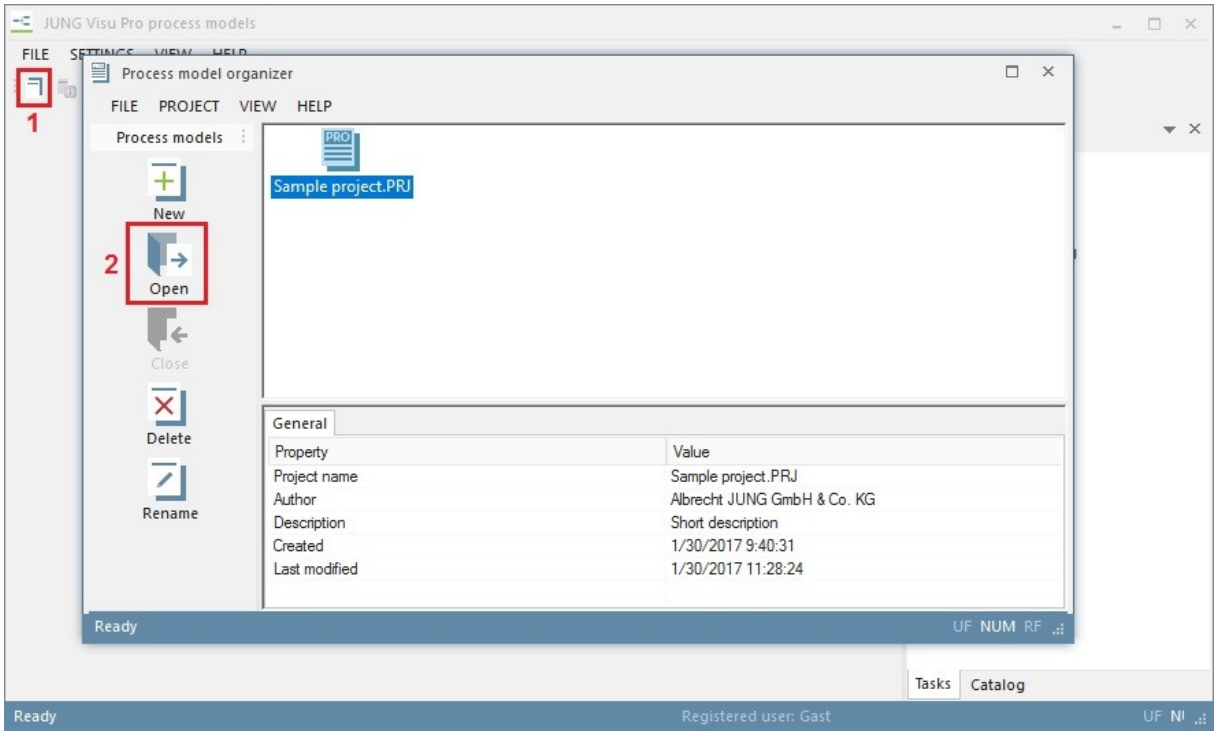
Author and description: These specifications are useful but not mandatory. They can be entered or changed later.



After entering at least the name of the project, click the button ***Create*** to create the new process model project. The new project appears in the project list of the project organizer.

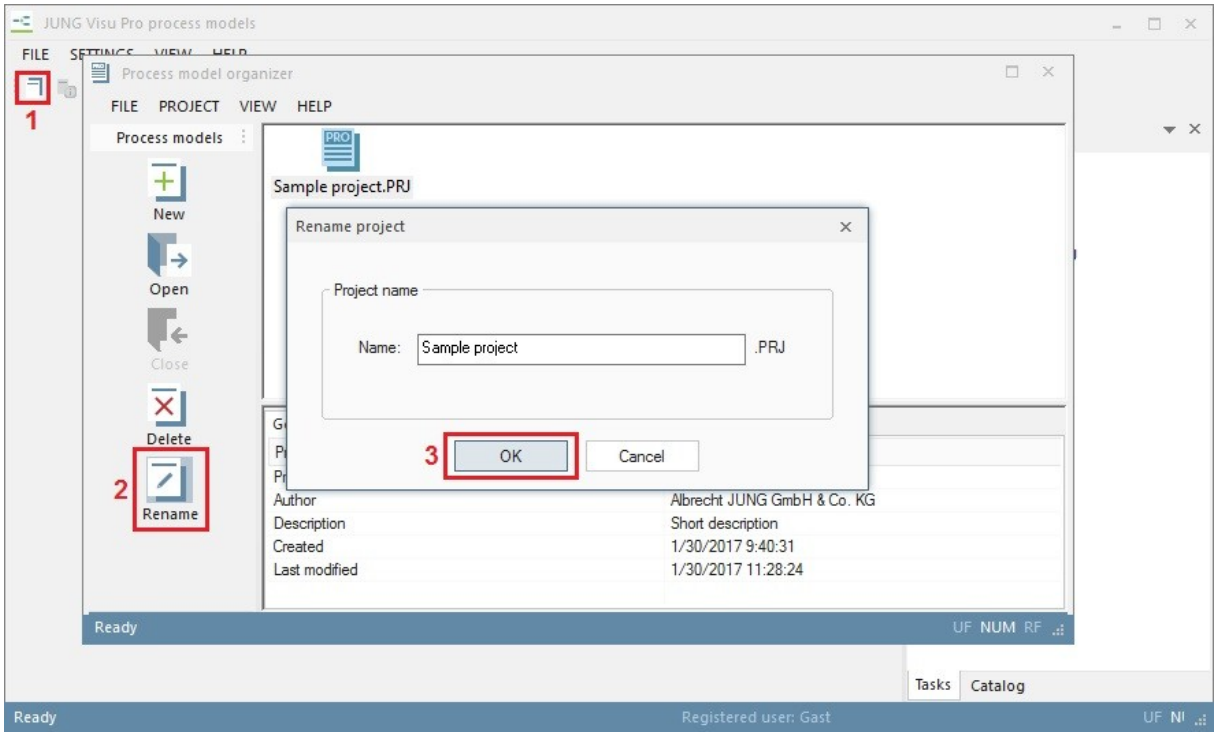
Open project:
To open the newly created project in the process model editor,

- double-click the project symbol or
- select the project and click the button ***Open*** in the sidebar or
- choose the menu item ***Project*** and click ***Open***.



Rename project:

To rename a project, select the desired project and click the button ***Rename*** in the sidebar. Alternatively, choose the menu item ***Project*** and click ***Rename***. The dialog ***Rename project*** opens.



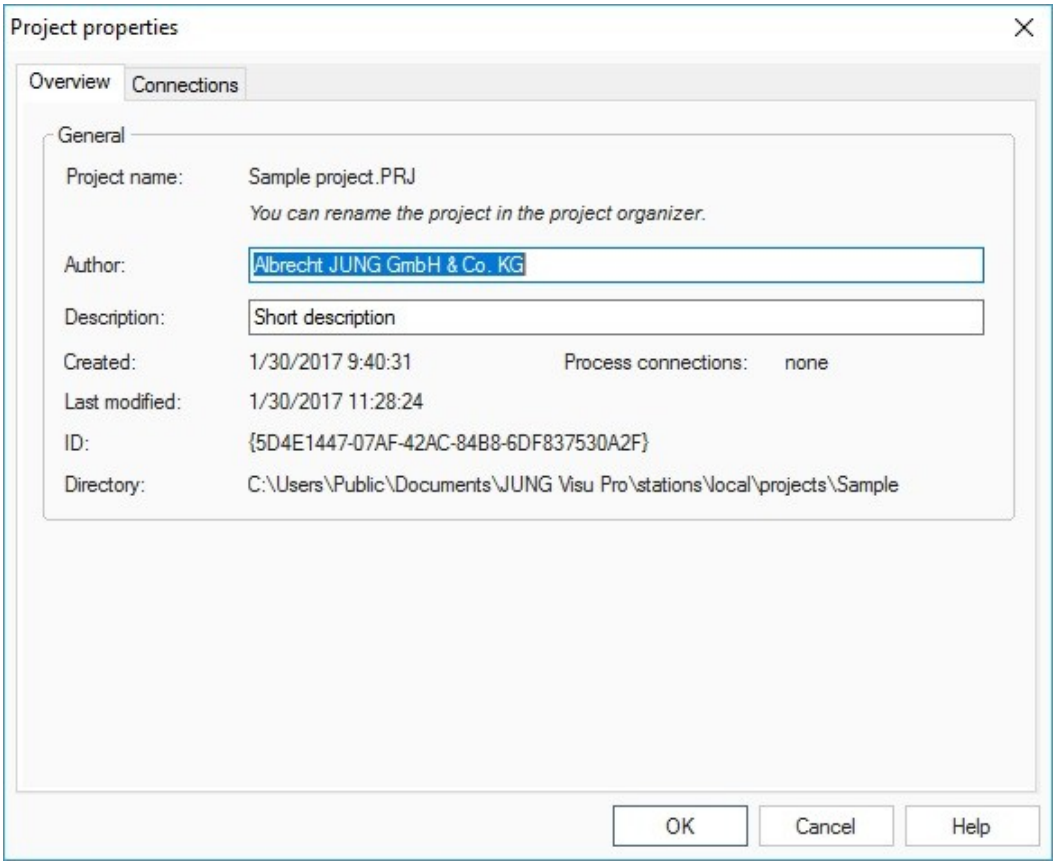
Enter the new project name and click the button ***OK***.

The dialog closes and the new project name is adopted. The extension ***.PRJ*** is added automatically to the project name.

Note that a project can only be renamed when it is not currently opened.

4.2 Project Properties

Choose in the process model editor the menu item ***File*** and click ***Project Properties*** or click the button in the toolbar. The dialog ***Project properties*** opens.



The tab ***Overview*** shows the general properties of the loaded project. Author and description of the project may be changed here.

After editing the properties click the button ***OK*** to commit the entries and to close the dialog.

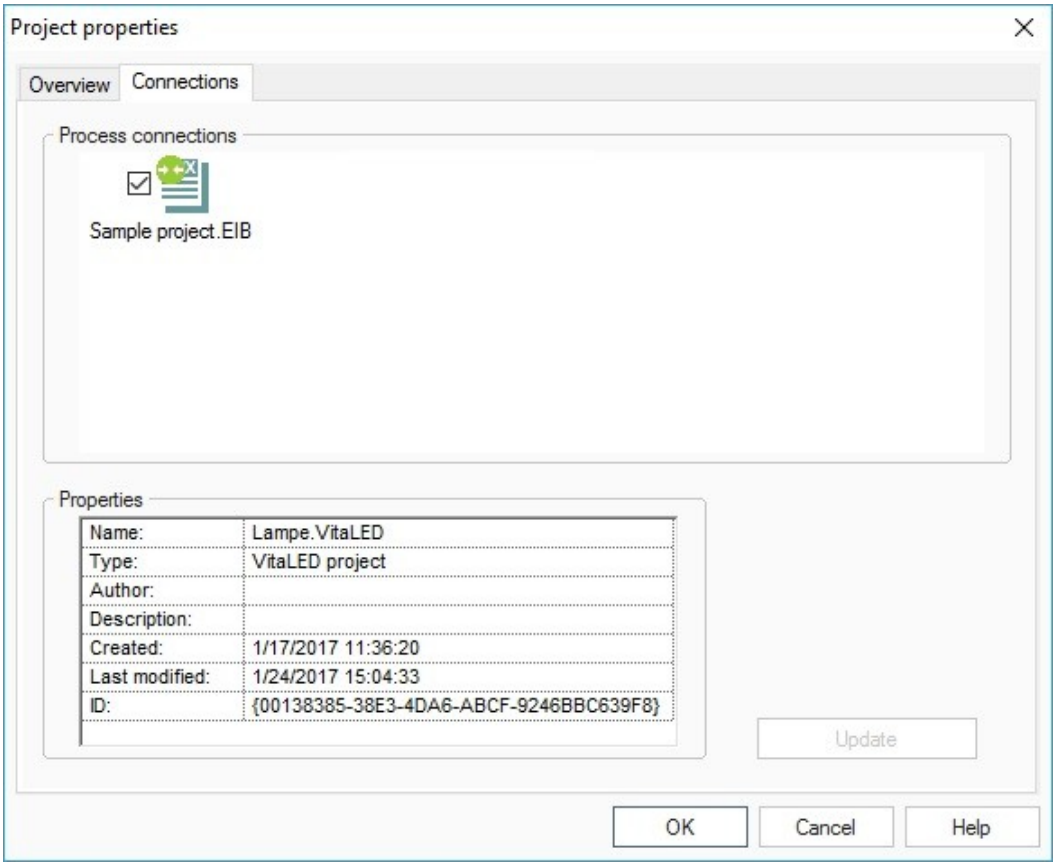
Note that the project name can be changed in the project organizer.

4.3 Import from Process Interface Projects

The process model must be connected to one or more process interface projects, i.e. KNX projects and OPC projects. The process variables in the process model will be created from the data points of the process interface projects.

The name, type and description of each process variable is initialized from the respective data point in the process interface project. A KNX group address of the type **Boolean**, for instance, will result in a **binary** process variable. Other KNX group addresses are mapped to analog or string process variables.

To import a process interface project into the process model, choose the menu item ***File*** and click ***Project properties*** or click the button in the toolbar. The dialog ***Project properties*** opens.



The tab ***Connections*** lists all available process interface projects.

The process interface projects used with the process model are marked by a check mark. To connect an interface project with the process model, select the checkbox at the left of the project symbol.

You can import more than one project, for example a KNX project for controlling the light and an OPC project for controlling the air conditioning system.

> ***Note:*** *It is possible to integrate both KNX and OPC projects in one process model.*

Click the button ***Update*** to update data in the process model from the selected process interface project. The selected projects are now loaded in the process model.

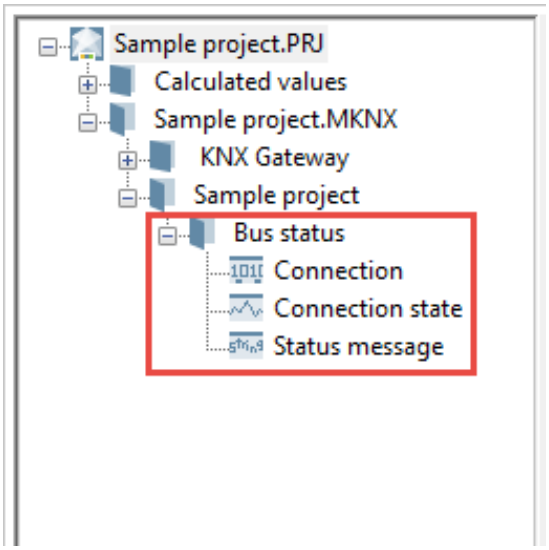
Click the button ***OK*** to commit your selection and to close the dialog.

The interface projects are now loaded and can be edited. The project structure of KNX projects remains the same as in the KNX editor.

On the left side of the editor a directory tree is displayed. It contains folders derived from the main and middle groups. The folders comprise the group addresses, now converted into process variables.

4.3.1 Folder Bus State

When importing a KNX project, a data point folder called bus state is created. This contains the following process variables:



The binary process variable *Connection* indicates whether a connection to the KNX interface exists.

0: No connection to the interface
1: Connection to the interface established

The analog process variable *Connection state* indicates whether the connection to the KNX interface was terminated correctly.

0: Connection established
1: Connection closed
2: Connection interrupted
4: Connection unknown

The process variable *Status message* of type String contains possible error messages of the FALCON driver.

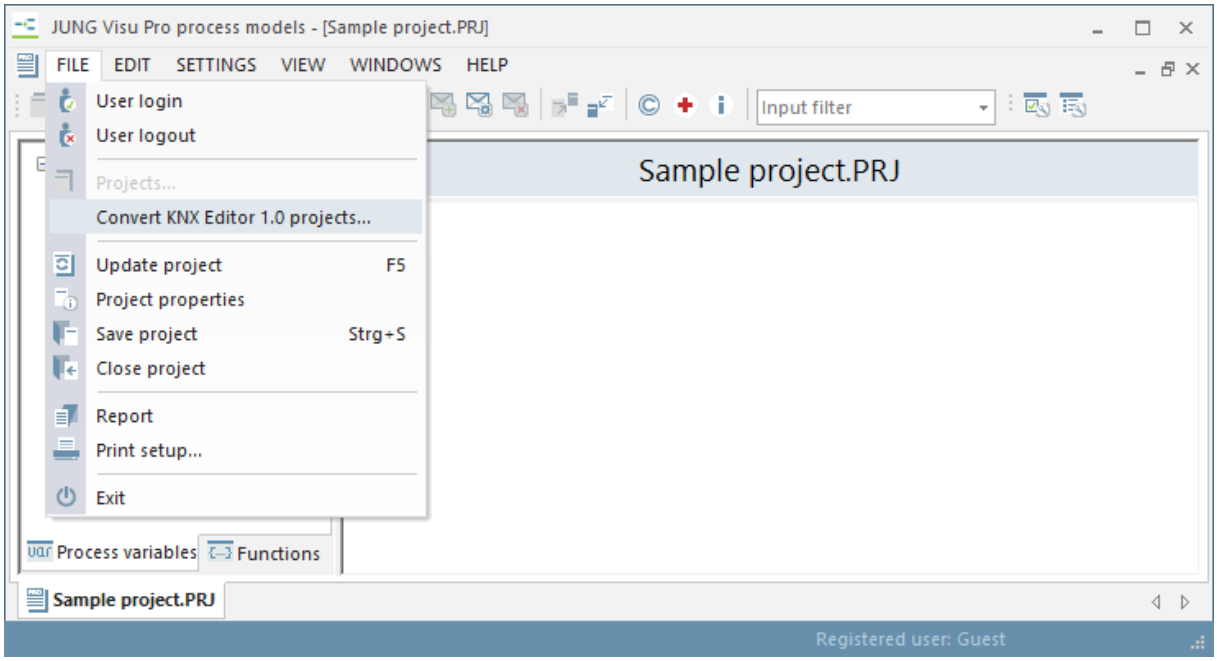
Note: The variables *Connection state* and *Status message* exist only in a KNX project 2.0.

4.4 Conversion of a KNX Project 1.0 to 2.0

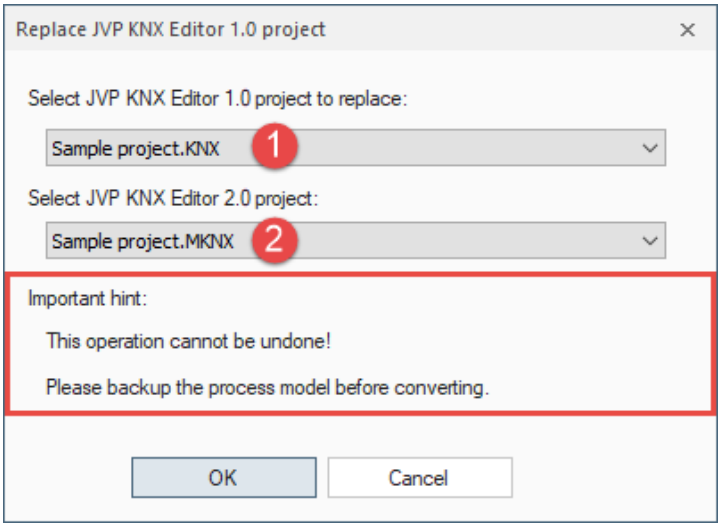
Beginning with version 4.4 there is an extended version of the KNX editor available in the JUNG Visu Pro, the KNX Editor 2.0. This allows a connection of several KNX projects in one editor. To use the new editor, however, a conversion of old projects is necessary. Converting is a two step process. First you have to create a new project in the KNX Editor 2.0 and convert your old KNX project 1.0 into it. Detailed instructions can be found in the KNX Editor 2.0 manual.

The second step is to update the process model. To import this project into the process model without having to reconnect the data points of your visualization, use the conversion function of the process model.

Choose the menu item ***File*** and click ***Convert KNX Editor 1.0 projects...***



A dialog opens in which you select your old KNX project 1.0 (1) and the newly converted KNX project 2.0 (2).



Attention:

- Please note the advice and save your process model beforehand, because the process is not reversible.
- Conversion is only possible if all group addresses from KNX project 1.0 are also contained in KNX project 2.0.

Confirm by clicking ***OK***.

4.5 Save the Process Model

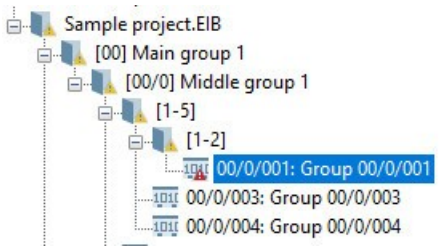
To save the process model, choose the menu item ***File*** and click ***Save project*** or click the button in the toolbar.

When the project or the process model editor are closed, the program saves automatically.

Now you can use the process model in the visualization.

4.6 Update from Process Interface Projects

The process model must be updated when a connected process interface project, i.e. a KNX project, has been changed. To update the process model, choose the menu item ***File*** and click ***Update project*** or press the function key ***F5*** to perform the update. Alternatively, the update from process interface projects can be made in the project properties dialog.



If a data point has been deleted in the process interface project, i.e. a KNX group address has been deleted in the KNX editor, the corresponding process variable will be displayed with a deletion mark.

You can decide whether you want to delete those variables in the process model as well. Prior to that you may check the results of the deletion.

To delete a process variable, select the process variable in the process variables structure tree and press the keyboard key ***Delete***.

Important:

*When the type of a data point has been changed in the process interface, i.e. a KNX data point has been changed from **Boolean** to **2 Octet Float**, the process variable of the old type will be marked as deleted and a new process variable with the new type will be created.*

The name, access rights, unit and description of a process variable will always be updated from the corresponding data point in the process interface project.

New data points in the process interface project will result in new process variables in the process model.

There is a special case with KNX projects: If the KNX group address scheme has been changed from a 2-level to a 3-level scheme or vice versa, the structure tree of the process variables will be changed accordingly. However, the process variables will be retained.

4.7 Properties of Process Variables

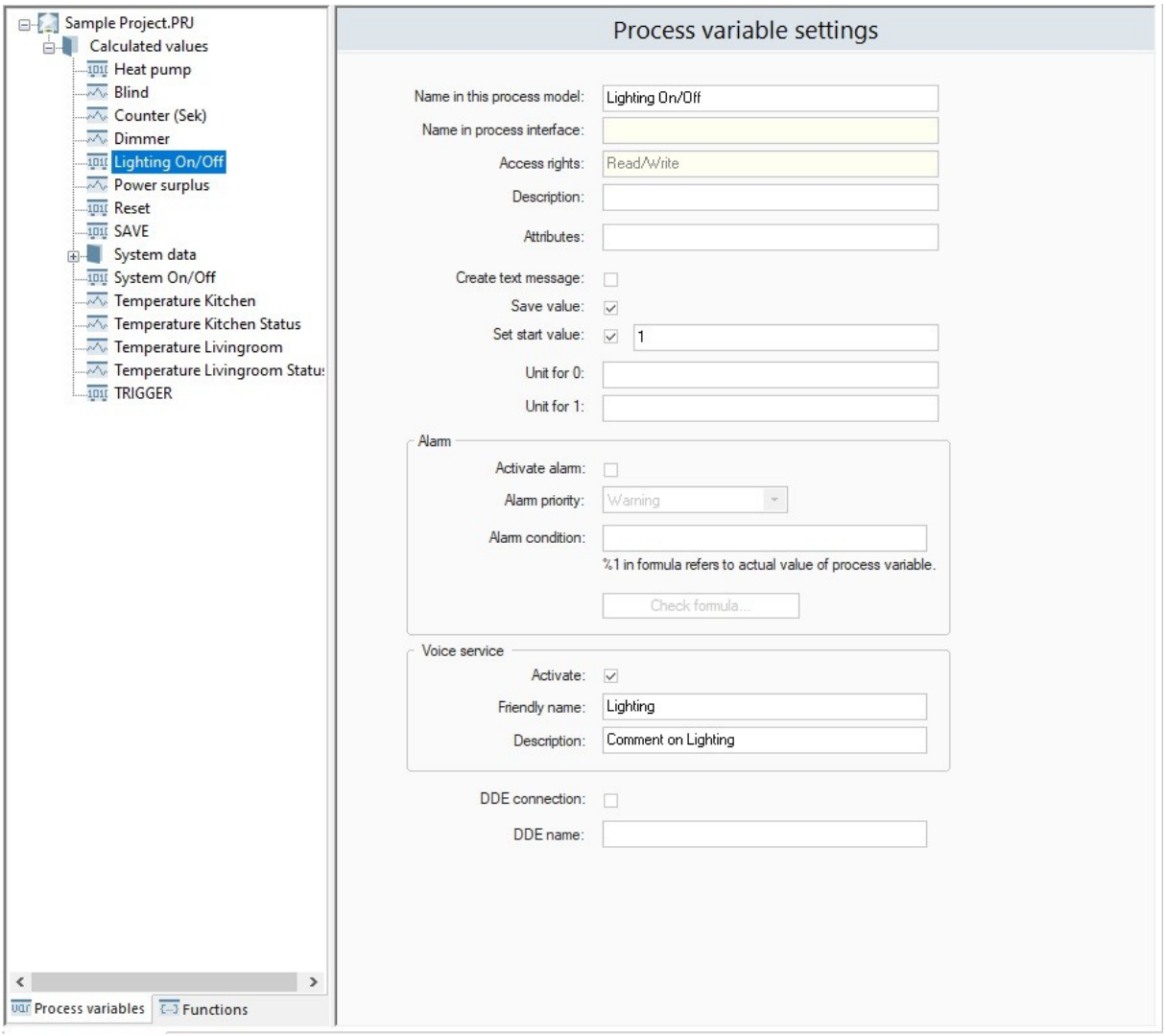
For each data point imported from a process interface project, i.e. a KNX project, a process variable is created in the process model.

The process variable in the process model is initialized from the data point in the respective process interface project. In addition to the properties copied from the data point, email notifications, archives, and other properties can be defined for the process variable.

To edit the properties of a process variable, select the process variable in the process variable structure tree. The properties of the process variable are displayed in the main window of the process model editor.

Properties of binary process variables:

> ***Note:*** *The mandatory properties of process variables were already initialized during the import of the process interface project. All changes to the process variable properties are optional.*



Name in the process model: The name of the process variable is initialized with the name of the data point in the process interface project, i.e. KNX group address in a KNX project. Here you can change it into a more meaningful name in the context of the application.

Description: The description is initialized with the description of the data point in the process interface project. You can edit it here.

Attributes: Attributes allowing you to give special properties to the variable, which influence the behavior of certain functions. Following attributes are available at the moment:

- Operation mode: Allows analog variables to be treated as operation modes in the time switch and the scene. Instead of a slider you can now switch between operation modes in the time switch/scene.

Create text messages: If enabled, for each change of the process variable value a text message will be created. Otherwise the generation of a text message will be suppressed.

Save value: If enabled, on termination of the process model, the value of the process variable will be saved and the process variable will be initialized with this value when the process model will be restarted.

Set start value: If enabled, the variable will be initialized with the defined value when the process model is started, unless a previously saved value for the process variable has been defined. A previously saved value will override the start value.

Unit for 0: Internally, binary values are encoded with the values 0 and 1. It is possible to assign a text to each numerical value, so that in plain messages the text will be used instead.

Unit for 1: See *Unit for 0*.

Activate alarm: If the value of a process variable changes, the process model will check an alarm condition, in case the option ***Activate alarm*** has been turned on. If the alarm condition holds true, a specified alarm handling will be started. In turn, the alarm will be signaled and for example an email message will be sent, if configured. All alarms and warnings are inserted into the system log file.

Alarm priority: Warnings are less severe alarms, depending on the application.

Alarm condition: The alarm condition is defined as a formula. Use "%1" (without quotes) in the formula to refer to the current value, which is either 0 or 1 for binary variables.

Example: *Signal alarm when the value is 1*

Formula: %1 = 1

Enable voice service: By activating the voice service, the process variable can be voice-controlled. It is necessary to specify a friendly name as well as a description. If this is not taken into account for each activated variable, no devices will be found.

Friendly name: The name, the process variable can for example be addressed with via Amazon Alexa:

Friendly name: Living room light
Command to Amazon Alexa: "Turn the living room light on"

Description: A description of the process variable that is displayed within the voice service interface.

DDE connection: If enabled, a WINDOWS DDE (Dynamic Data Exchange) connection with read access will be set up for the process variable.

DDE name: DDE item name used to read the value of the process variable from other applications, i.e. Microsoft Excel.

To address the process variable from DDE, a complete specification of the process variable is composed of the name of the DDE server ("iPhon DDE Server"), the name of the process model and the DDE item name of the process variable.

Excel example:

DDE name (DDE item):	room_temperature
Process model (DDE topic):	school.prj
iPhon DDE Server (DDE server):	iPhon DDE Server

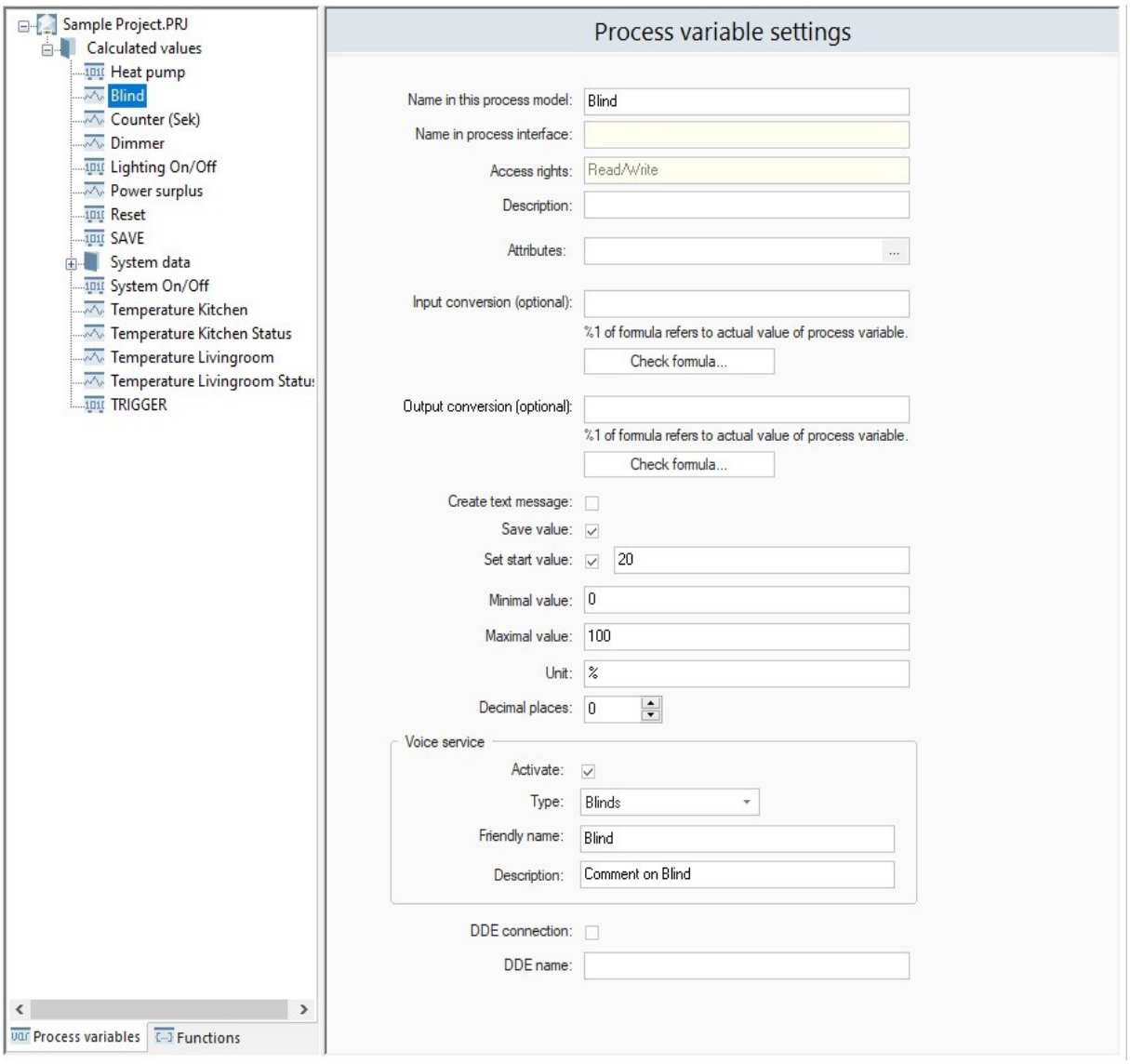
The name of the DDE variable in Excel:
'iPhon DDE Server'|school.prj!room_temperature

Excel requires names quoted with single quotation marks, if spaces are used.

Access rights and name in the process interface project must be modified in the process interface project and cannot be edited here.

Properties of analog process variables:

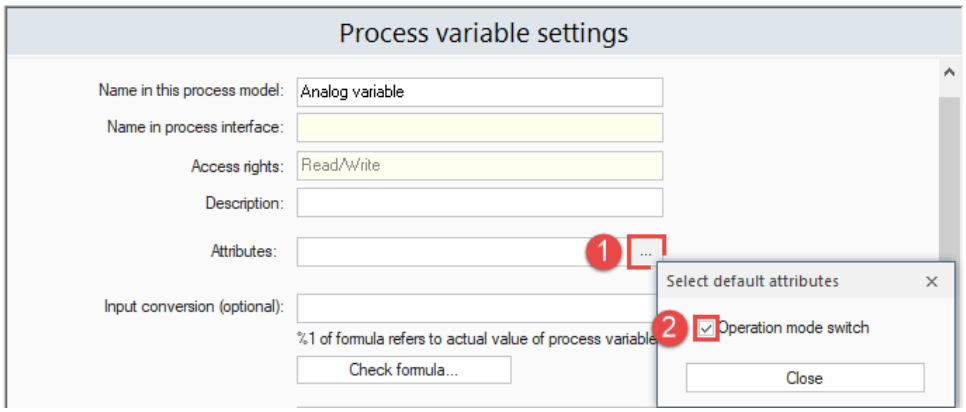
> ***Note:*** *The mandatory properties of process variables were already initialized during the import of the process interface project. All changes to the process variable properties are optional.*



Name in the process model: The name of the process variable is initialized with the name of the data point in the process interface project, i.e. KNX group address in a KNX project. Here it can be changed to a more meaningful name in the context of the application.

Description: The description is initialized with the description of the data point in the process interface project. Here it can be changed.

Attributes: Attributes allowing you to assign special properties to a variable, which influence the behavior of certain functions. A click on the 3 dots (1) opens a selection dialog for existing default attributes (2).



Following attributes are available at the moment:

- Operation mode: Allows analog variables to be treated as operation modes in the time switch and the scene. Instead of a slider you can now switch between operation modes in the time switch/scene.

Input conversion: The value received from the process interface does not always have the proper type from an application's point of view. For example a temperature sensor may return a voltage which must be converted to a temperature.

The conversion can be described with a formula. Use "%1" (without quotes) in the formula to refer to the actual value.

Operators: +, -, ^, *, /, Mod, +, -, &, =, <>, <, >, <=, >=, Not, And, Or, Xor. Further details in chapter 6.5.

You may also use parenthesis.

> ***Example:*** *The sensor sends a voltage instead of a temperature. The range 0...10 V must be depicted to 0...15 °C.*
>
> **Formula:** %1 * 1.5

Output conversion: The value can be converted before it will be sent, similar to the conversion applied to values received.

Check formula: The button ***Check formula*** opens a dialog which will immediately show the result of a formula applied to different input values.

Enable voice service: By activating the voice service, the process variable can be voice-controlled. It is necessary to specify a friendly name as well as a description. If this is not taken into account for each activated variable, no devices will be found.

Friendly name: The name, the process variable can for example be addressed with via Amazon Alexa:

Friendly name: Living room temperature
Command to Amazon Alexa: "Set the living room temperature to 21 degrees "

Description: A description of the process variable that is displayed within the voice service interface.

Minimal value: The process model will not send values below the minimal value. This option can be used to ensure proper operation of devices.

Maximal value: The process model will not send values above the maximal value. This option can be used to ensure proper operation of devices.

Unit: If defined, the text of the unit will be used in plain messages.

Decimal places: Number of decimal places used for plain messages.

DDE connection: If enabled, a WINDOWS DDE (Dynamic Data Exchange) connection with read access will be set up for the process variable.

DDE name: DDE item name used to read the value of the process variable from other applications, i.e. Microsoft Excel. See binary process variables as an example.

4.8 Calculated Process Variables

Function description:

In addition to process variables created from data points in process interface projects, i.e. KNX or OPC projects, new process variables can also be defined by the user. The values of self-defined process variables are usually calculated from already existing process variables. As an exception, the value of a self-defined process variable may not be calculated based on other process variables, but can be controlled for example by a calendar program. This is regarded as a special case of calculated process variables.

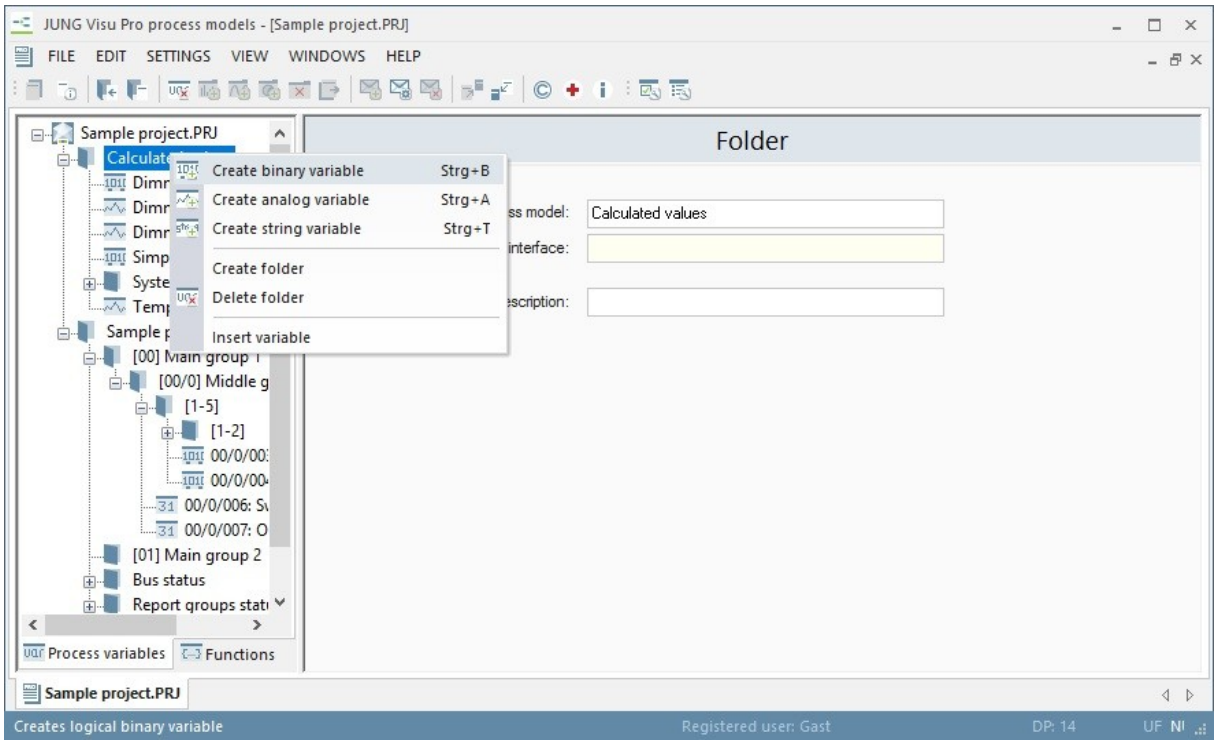
Example: *Counter*

The process interface returns pulse information, i.e. from an electricity counter, each pulse representing a certain energy consumption. The process model provides functions which allow to count the pulses and to add up the consumption values. In this case, a new process variable will be defined to hold the total consumption.

The process model provides the same features for self-defined process variables as for process variables derived from data points in process interface projects. For example, you may create archives of self-defined variable values and can apply the alarm handling.

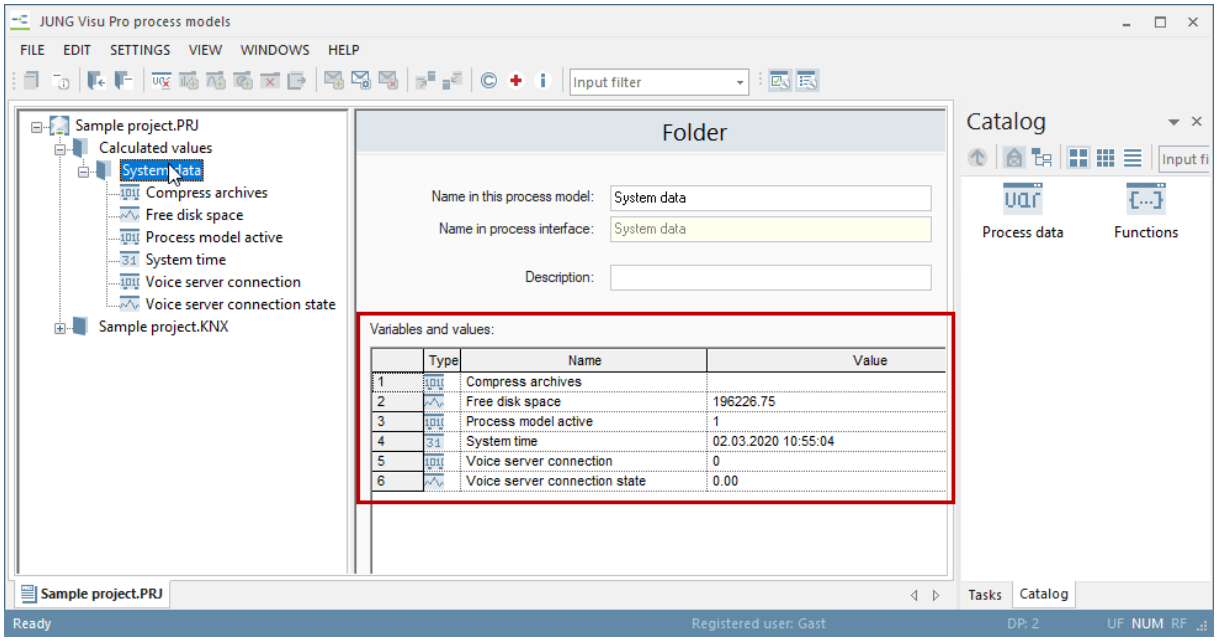
Creation of self-defined process variables:

To define a process variable, right-click the folder ***Calculated values*** to open the context menu. Then select the type of process variable to be created.



Then click on ***Calculated Values*** folder in the directory tree and there on the variable just created to configure the settings for the process variable in the middle of the editor.

If you click on a folder, the current values of all process variables located in this folder are displayed.



5 Alarm Limits

Function description:

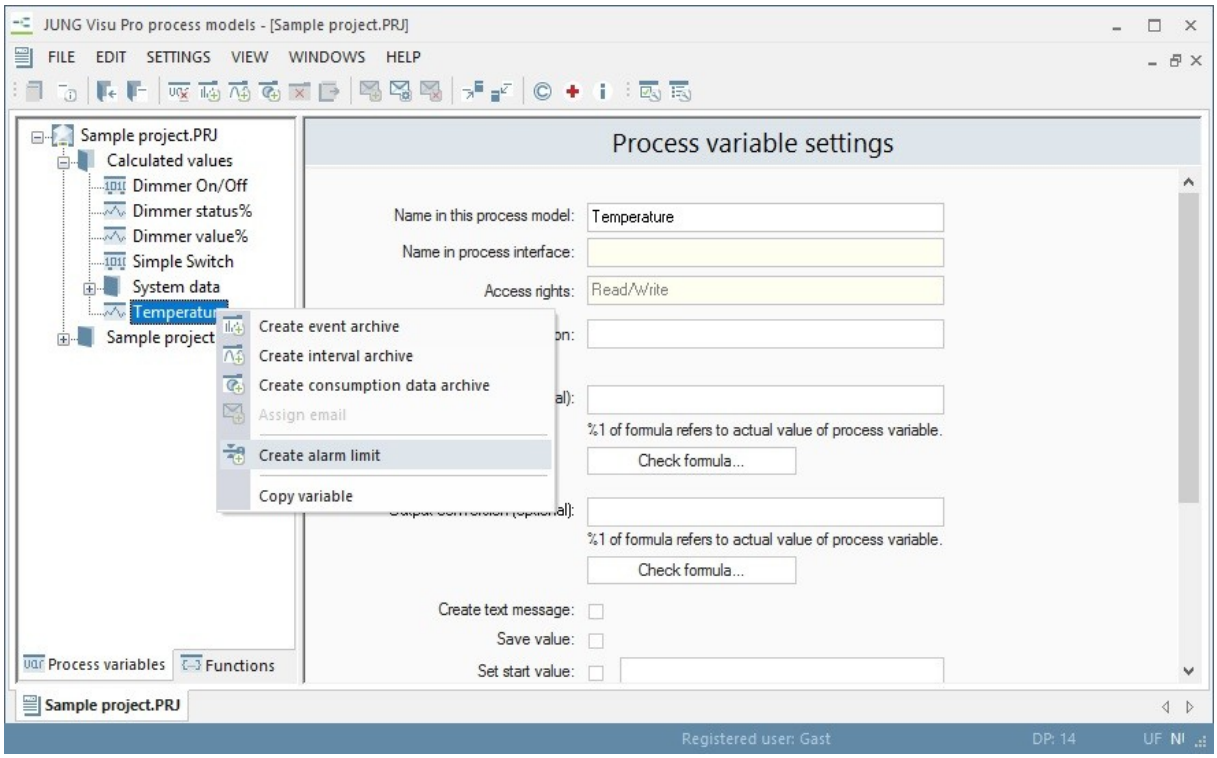
Binary process variables can simply define alarms with an optional alarm condition. The alarm condition is defined by means of a formula based on the value of the binary process variable. Since the value of a binary variable is either 0 or 1, there is at most one alarm condition.

Analog process variables may require several warnings and alarms, i.e. a warning when the temperature reaches 5° Celsius and an alarm when the temperature reaches 10° Celsius.

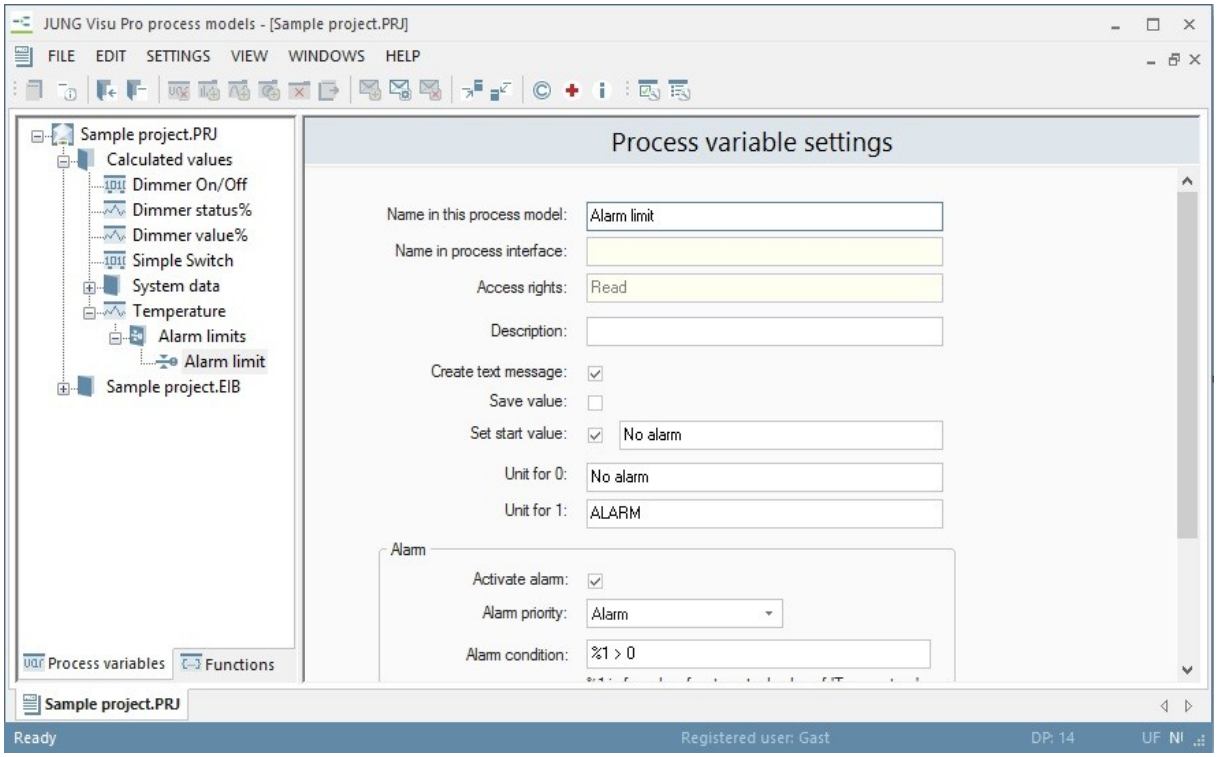
For each alarm condition an alarm limit can be defined, which has value 1 if the condition of the analog variable holds true and value 0 in the opposite case. The condition is defined by means of a formula. These alarm limits are quite similar to regular binary variables, the most important difference is the alarm condition: The alarm condition does not refer to the alarm limit's value but to the analog variable to which it has been attached.

Create alarm limit:

Right-click an analog process variable to open the context menu. Then click ***Create alarm limit***. Beneath the analog process variable an ***Alarm limits*** subfolder will be created which already comprises a first alarm limit.



Select the alarm limit to define its properties.



Name in the process model: The name of the alarm limit.

Description: The description of the alarm limit.

Create text messages: If enabled, for each change of the alarm limit's value a text message will be created. Otherwise the generation of a text message will be suppressed.

Save value: If enabled, on termination of the process model the value of the process value will be saved and the process variable will be initialized with this value when the process model will be restarted.

Set start value: If enabled, the variable will be initialized with the defined value, when the process model is started, unless a previously saved value for the process variable has been defined. A previously saved value will override the start value.

Unit for 0: Internally, binary values are encoded with the values 0 and 1. It is possible to assign a text to each numerical value, so that in plain messages the text will be used instead.

Unit for 1: See *Unit for 0*.

Activate alarm: If the value of a process variable changes, the process model will check an alarm condition, in case the option ***Activate alarm*** has been enabled. If the alarm condition holds true, a specified alarm handling will be started. In turn, the alarm will be signaled and for example an email message will be sent, if configured. All alarms and warnings are inserted into the system log file.

Alarm priority: Warnings are less severe alarms, depending on the application.

Alarm condition: The alarm condition is defined as a formula. Use "%1" (without quotes) in the formula to refer to the current value of the analog variable.

> ***Example:*** *Signal alarm when the analog variable value is above 0.*
>
> Formula: %1 > 0

DDE connection: If enabled, a WINDOWS DDE (Dynamic Data Exchange) connection with read access will be set up for the alarm limit.

DDE name: DDE item name used to read the value of the alarm limit from other applications, i.e. Microsoft Excel.

To address the process variable from DDE, a complete specification of the process variable is composed of the name of the DDE server ("iPhon DDE Server"), the name of the process model and the DDE item name of the process variable.
See properties of binary process variables as an example.

6 Email Notifications

Email notifications can be assigned to binary process variables and to alarm limits.

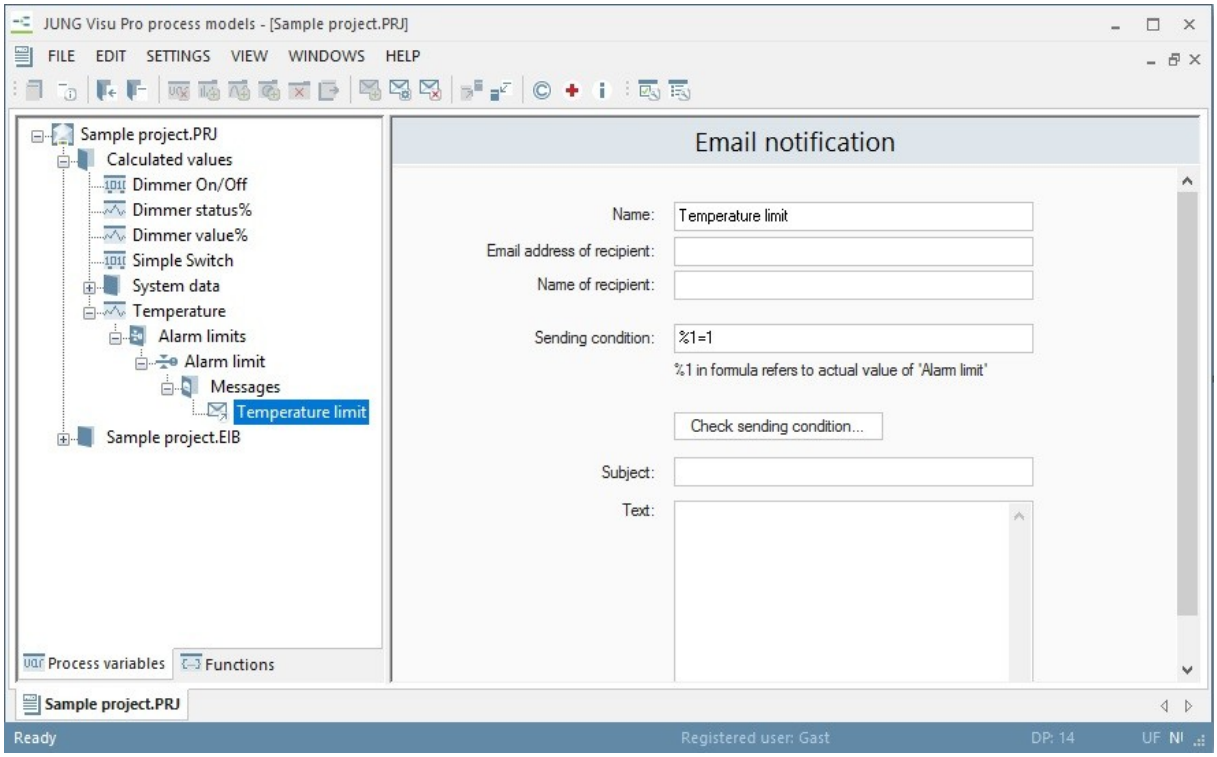
Each email notification comprises its own condition. Whenever the value of a binary process variable or an alarm limit changes, the process model checks whether the predefined condition is met and an email notification has to be sent. The condition to send an email notification is not identical with an alarm condition: Alarm personnel may not only be informed when an alarm condition becomes true, but also when later the alarm condition becomes false.

> ***Note:*** *Email notifications can only be used if the service for sending emails is installed on the computer.*

6.1 Create Email Notification

Select a process variable. Choose the menu item ***Edit*** and click ***Assign email*** or click the button in the toolbar to create a new email notification for the process variable.

The subfolder ***Notifications*** will be created in the folder of the process variable with an empty email notification already defined. To configure the newly created email notification, click it. Its properties will be displayed in the main window of the process model editor.



Name: Name of the email notification. This name will be used in the structure tree to identify the email notification.

Email address of recipient: Enter the recipient's email address.

Name of recipient: Enter the name of the recipient.

Sending condition: A formula is used to specify the sending condition. The email will be sent when the formula returns true.

Use "%1" (without quotes) in the formula to refer to the current value of the process variable or the alarm limit.

Operators: +, -, ^, *, /, Mod, +, -, &, =, <>, <, >, <=, >=, Not, And, Or, Xor. Further details in chapter 6.5.

You may also use parenthesis.

Example: An email is to be sent if the reported value of the binary process variable is ***1*** or if the value of the alarm limit is ***1*** (in the case of an analog variable with an assigned email notification).

The condition is: %1 = 1

Subject: Enter the email subject.

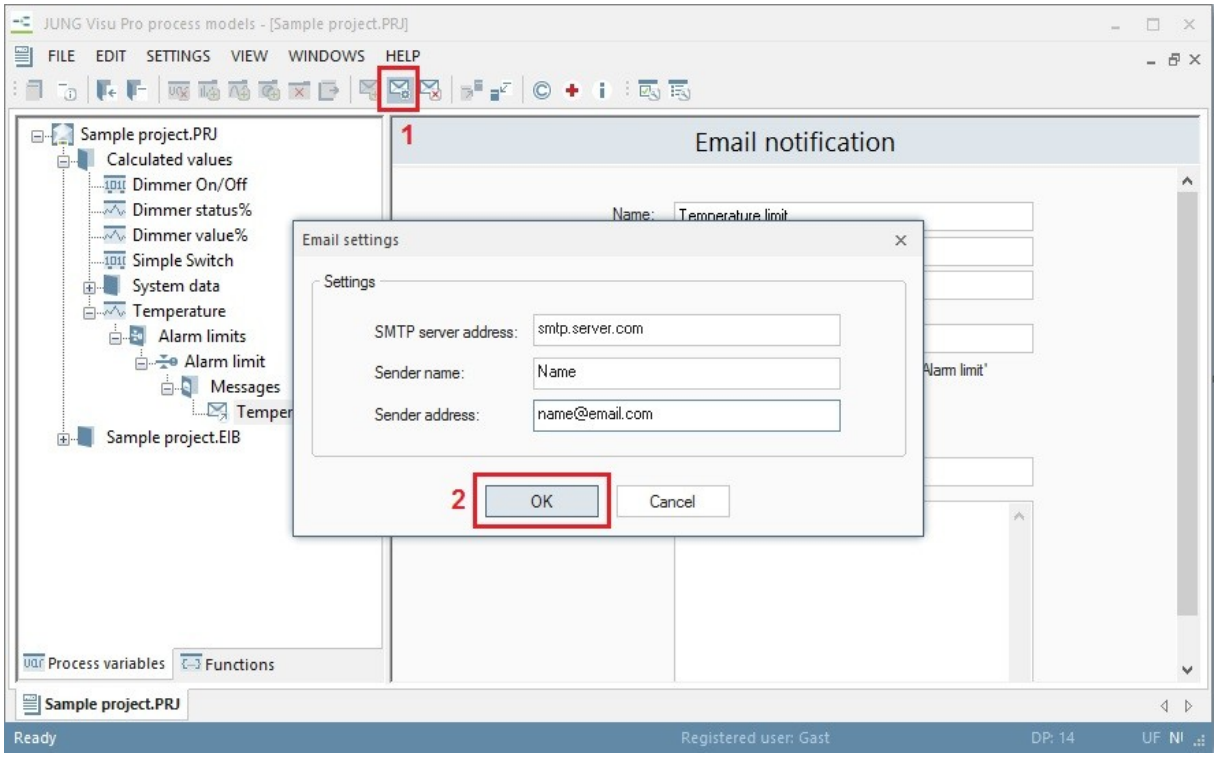
Text: Enter the text of the email. Use "%1" (without quotes) in the formula to refer to the actual value of the process variable or the alarm limit.

6.2 Delete Email Notification

To delete an email notification, choose the menu option ***Edit*** and click ***Delete email*** or click the button in the toolbar. Alternatively, press the keyboard key ***Delete***.

6.3 General Email Settings

In order to send emails, the process model requires some general information. To configure general email settings, choose the menu item ***Settings*** and click ***Email configuration*** or click the button in the toolbar.

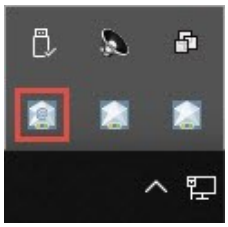


Enter in the dialog ***Email settings*** the following information:

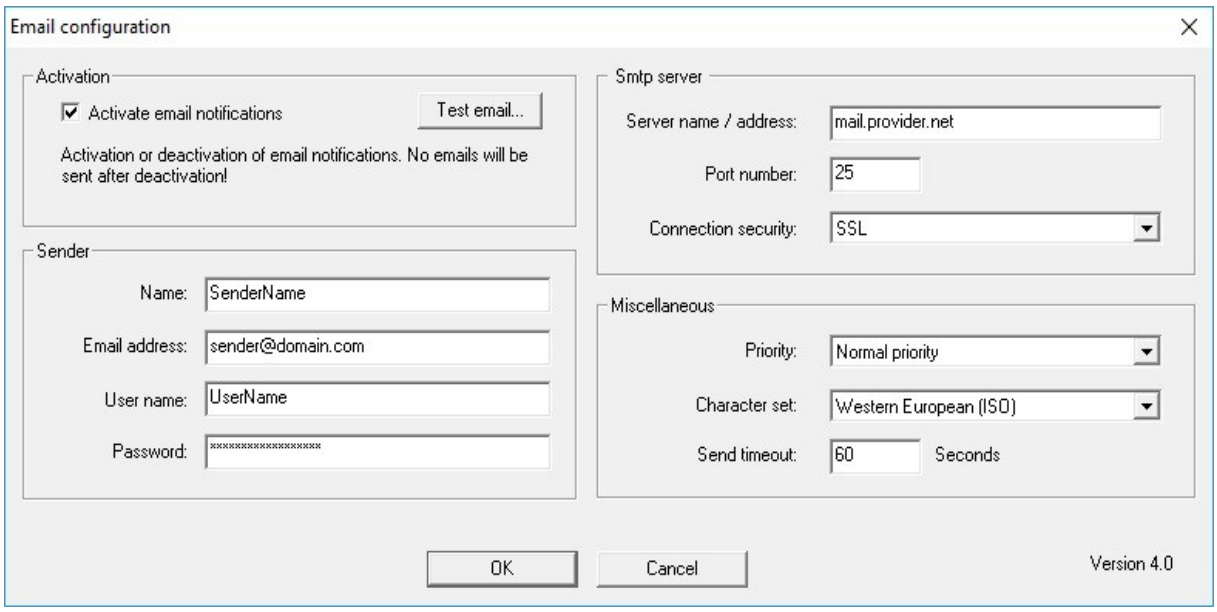
SMTP server address: Enter the address of the SMTP server.
Sender name: Enter the name of the sender.
Sender address: Enter the email address of the sender.

6.4 Extended Email Settings

The email client can be configured directly in the user interface of the email client. The email client is visible in the Windows status bar as a small icon.



Double-click the icon with the email symbol to open the dialog ***Email configuration***.



Enter the following information:

Activation

Activate email notifications: Activates or deactivates the email client.

Note: Emails cannot be sent if the email client is inactive!

All changes of the activation state will be recorded in the system logfile. On system start the activation state of the email client will be recorded in the logfile.

Test email: Opens the dialog to send an email for testing.
To send the test mail, the email client uses the same settings which are visible in the dialog.

Sender

Name: Name of the sender.

Email address: Email address of the sender. This field is mandatory.

On email reception the sender identification will be composed of the sender name and the email address of the sender, e.g.

John Testman <j.testman@domain.com>

If the name of the sender is not specified, the email address will be used as sender identification.

User name: The user name which will be used by the email client to log in to the smtp server. The user name will be provided by the email service provider.

Password: The password which will be used by the email client to log in to the smtp server. The password will be provided by the email service provider.

Smtp server

Server name/address: Address of the smtp server.

Port number: Port number of the smtp service of the smtp server. The default port number is 25.

Connection Security: Choose the type, that should be used. Following options are available: **SSL**, **STARTTLS** and **none**. The default option is **none**.

Miscellaneous

Priority: Priority of the email.

Character set: Specifies the character set to encode the email message.

Send timeout: Specifies the time span for the client to wait for answers from the mail server after the connection has been established.

The last 25 send attempts will be archived in protocol files. These send logs will be stored in the zip-archive '*CallServerLog.zip*' in the 'Temp'-directory of the visualization.

Note: The exact settings depend on the e-mail provider and can be found in their settings.

The following settings apply to Gmail, for example:

- in "User name" you have to enter the e-mail address
- the port is 465
- as "Connection security" choose "SSL"
- furthermore, "Less secure app access" must be allowed in the e-mail account

6.5 Checking Formulas and Conditions

Formulas are used as a general means to define conditions.

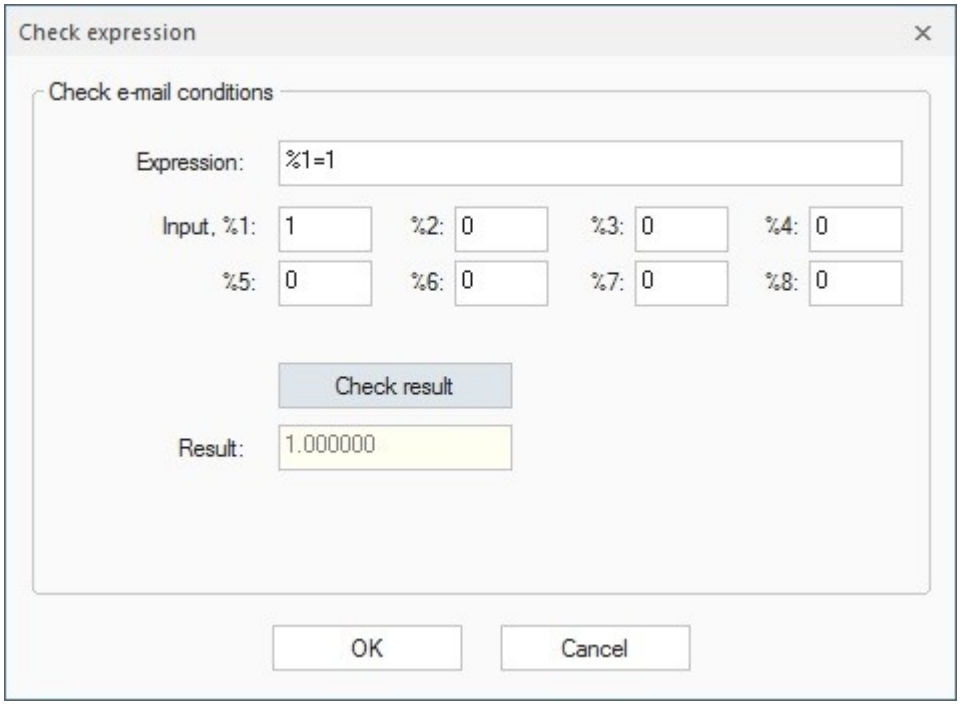
Operators: **+, -, ^, *, /, Mod, +, -, &, =, <>, <, >, <=, >=, Not, And, Or, Xor**

You may also use parenthesis.

Different dialogs allow to enter formulas and also provide the possibility to check the formula before it will be used.

The formula checking dialog allows to enter a mathematical expression, which refers to one or more input variables via "%1", "%2",.., the sign "%" followed by the number of the input variable.

To check the formula, click the button ***Check formula***. The dialog ***Check expression*** opens. Arbitrary values can be tested here. For real numbers, the period is used to separate the decimal places.



Click the button ***Check result*** to check the entered formula. You can correct the formula in the dialog.

Expression: In this field you can edit the formula.

Following operators can be used:

+ , - , * , /	Addition, Subtraction, Multiplication, Division
<>	Returns 1 if both values are not equal, else 0
=	Returns 1 if both values are equal, else 0 If you want to match two strings, the strings have to be written with single quotation marks ' . Example: %1 = 'Test'
MOD(%n,%m)	Calculates the remainder of the division %n/%m
<,>	Less, greater. Returns 1 if the first value is less (greater) than the second value, else 0.
<=, >=	Less equal, greater equal. Returns 1 if the first value is less(greater) than or equal to the second value, else 0
AND, &	Returns 1 if the first value and the second value are bigger than 0, else 0
OR	Returns 1 if the first value and/or the second value are bigger than 0, else 0
XOR	Returns 1 if the first value or the second value are bigger than 0, else 0
NOT(%n)	Returns 1 if %n=0, else 0

Input: Enter here the simulated value and click the button ***Check formula***. When entering analog values, make sure to use periods and not commas as delimiters.

Result: In this field the result of the calculation is displayed. The result cannot be edited.

7 Archives

For each process variable one or more archives can be created. The data of the archives are saved in the respective project folder in the file "EsfPVArchive.mdb".

Event archive: In an event archive each change of the process variable value will be stored.

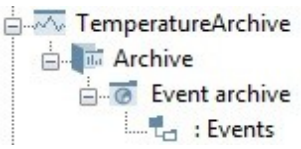
Interval archive: In an interval archive the archived values are computed based on a defined time interval, i.e. 5 minutes, 1 hour,

Consumption data archive: This archive stores consumption data.

> ***Limits:***
>
> *The number of archives is not limited by the process model editor. However, there are reasonable limits based on the available disk space, the frequency of value changes and the computation power of the system.*

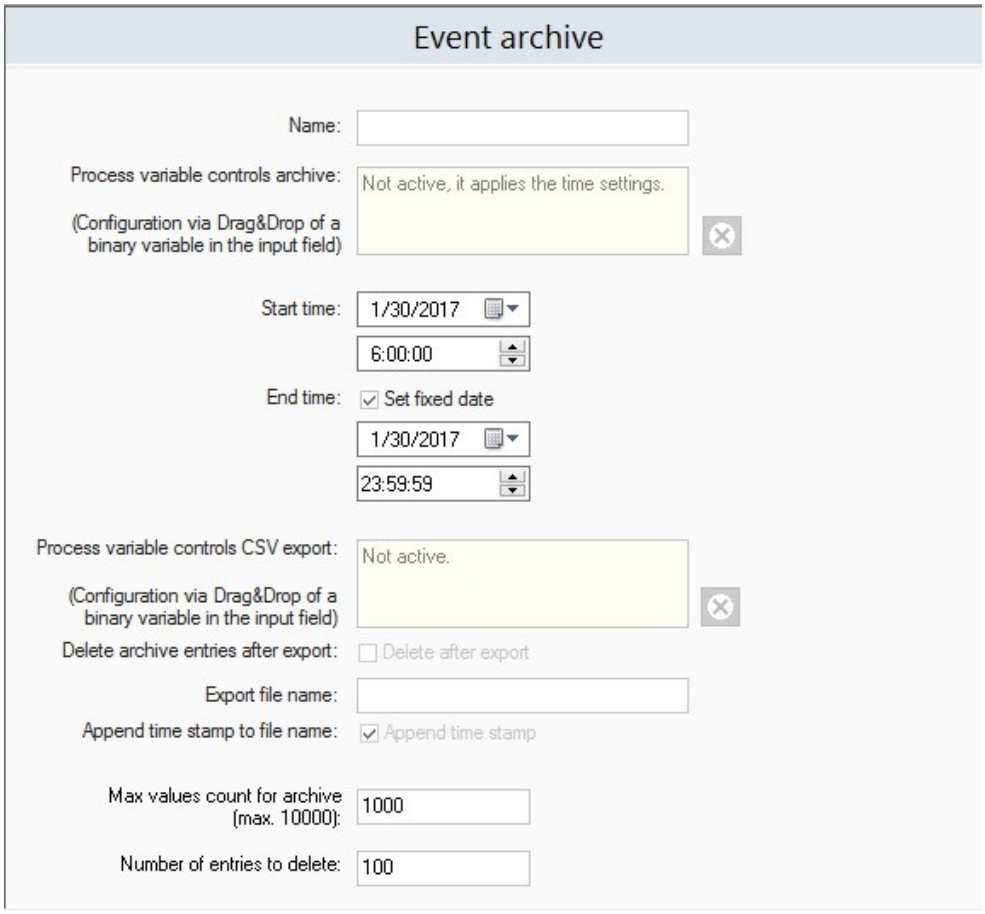
7.1 Event Archives

Every change of the value of the selected process variable is registered in the archive.



Create event archive:

Select the process variable for which you want to create the event archive. Choose the menu item ***Edit*** and click ***Create event archive*** or click the button in the toolbar. The newly created archive will be assigned to the selected process variable.



Enter the following information:

Name: Enter the name of the archive.

Process variable controls archive: This option allows to enter a process variable that controls the activation of the archive. If the value of the process variable is 1, the archive is activated and values are stored in the archive. If the value is 0, the archive is inactive and values will not be stored in the archive.

This option allows you to configure time controlled archives, which are able to collect values only on weekends (using the calendar application of the visualization system) or archives which will be active on various conditions. Important: the trigger event will not be stored in the archive.

Start time: Enter the date and time at which the value archiving should start.

End time: Enter the date and time at which the value archiving should stop.

If no end time has been specified, the archive will be organized as a first-in-first-out storage. If the maximal number of archived values has been reached, the oldest entries will be deleted to reclaim space in the archive.

Process variable controls CSV data export: This option allows to enter a process variable that will trigger the creation of a CSV formatted file containing the archived values. If the value of this process variable changes from 0 to 1, all archived values will be written into a CSV file. This option allows you to create CSV files with the archived values weekly or monthly.

Delete archive entries after export: Specifies whether the archived data should be deleted after creation of the export file.

Export file name: Name of the CSV file to create.

Append time stamp to file name: Specifies whether the name of the CSV file shall be extended by a time stamp (e.g. to prevent an existing CSV file from overwriting).

Max values count for archive: Maximal number of values in this archive.

Number of entries to delete: Deleting entries in the archive to reclaim storage will be more efficient if a high number of entries can be deleted at once. Enter the maximal number of entries the archive is allowed to delete in order to reclaim storage.

Delete event archive

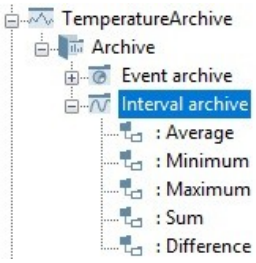
To delete an event archive, first select the archive in the structure tree. Then choose the menu option ***Edit*** and click ***Delete archive*** or click the button in the toolbar. Alternatively, press the keyboard key ***Delete***.

7.2 Interval Archives

In an interval archive the archived values are computed based on a defined time interval, i.e. 5 minutes, 1 hour,

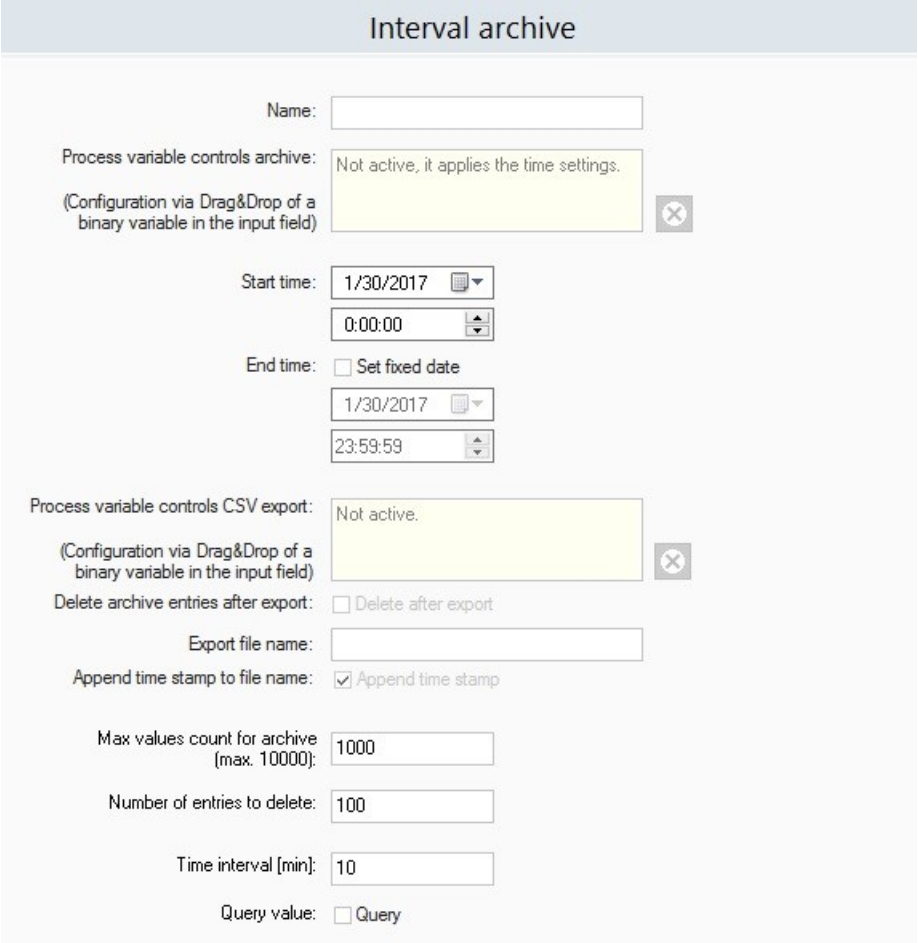
For each interval the archive will store several computed values:

- Average of all values received.
- Minimum of all values received.
- Maximum of all values received.
- Total sum of all values received (useful for calculation of consumption values).
- Difference of total sum to respective value of previous interval (useful for calculation of consumption values).



Create interval archive:

Select the process variable in the structure tree, choose the menu item ***Edit*** and click ***Create interval archive*** or click the button in the toolbar. The newly created archive will be assigned to the selected process variable.



Enter the following information:

Name: Enter the name of the archive.

Start time: Enter the date and the time at which value archiving should start.

End time: Enter the date and the time at which value archiving should stop.

If no end time has been specified, the archive will be organized as a first-in-first-out storage. If the maximal number of archived values has been reached, the oldest entries will be deleted to reclaim space in the archive.

Max values count for archive: Maximal number of values in this archive.

Number of entries to delete: Deleting entries in the archive to reclaim storage will be more efficient if a high number of entries can be deleted at once. Enter the maximal number of entries the archive is allowed to delete in order to reclaim storage.

Time interval: Enter the interval in minutes.

Query value: If enabled, within each interval the value of the process variable will be queried. This option avoids intervals with no value.

> ***Note:***
> *You may define several archives with different time intervals for one process variable.*

Example: Archiving temperatures

There are two interval archives for a temperature process variable:

One archive with
- Time interval: 10 min
- Maximum entries: 144 entries
- Query values: Yes

This archive will store 10 minutes average values for the last 24 hours.

One archive with
- Time interval: 60 min
- Maximum entries: 720
- Query values: No (values are already queried)

This archive will store hourly average values for the last 30 days.

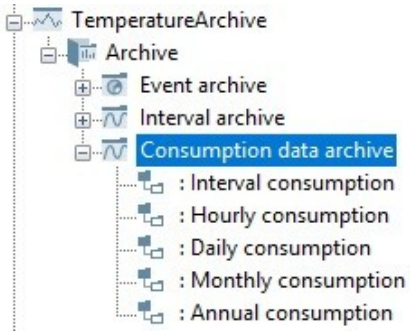
Delete interval archive

To delete an interval archive, select the archive in the structure tree. Then choose the menu item ***Edit*** and click ***Delete archive*** or click the button in the toolbar. Alternatively, press the keyboard key ***Delete***.

7.3 Consumption Data Archives

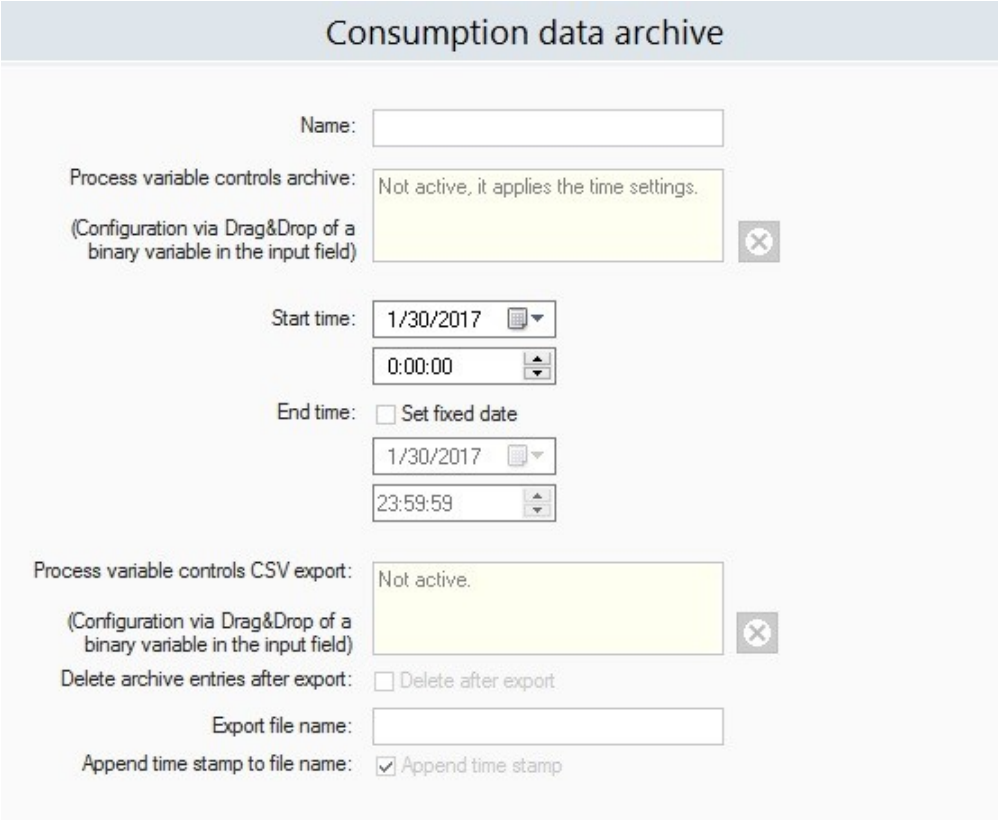
The consumption data archive stores a calculated value for a defined time interval. The following data are collected for the consumption:

- Interval consumption
- Hourly consumption
- Daily consumption
- Monthly consumption
- Annual consumption



Create consumption data archive:

Select the process variable for which you want to create the archive. Then choose the menu item ***Edit*** and click ***Create consumption data archive***. The newly created archive is assigned to the selected variable.



Enter the following information:

Name: Enter the name of the archive.

Start time: Enter the date and time at which the value archiving should start.

End time: Enter the date and time at which the value archiving should stop.

If no end time has been specified, the archive will be organized as a first-in-first-out storage. If the maximal number of archived values has been reached, the oldest entries will be deleted to reclaim space in the archive.

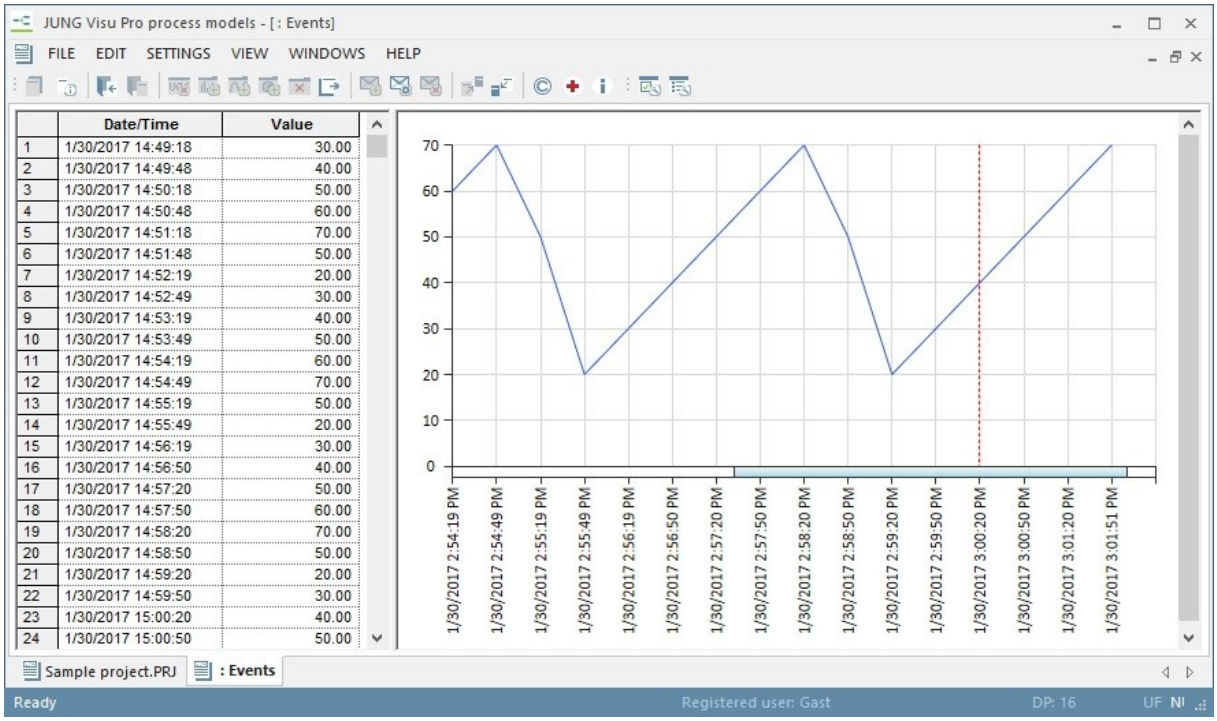
Delete consumption data archive

To delete a consumption data archive, select the archive in the structure tree. Then choose the menu option ***Edit*** and click ***Delete archive*** or click the button in the toolbar. Alternatively, press the keyboard key ***Delete***.

7.4 View Archives

The process model editor provides a viewer for archived values.

To open the viewer, double-click the archive in the process variables structure tree.



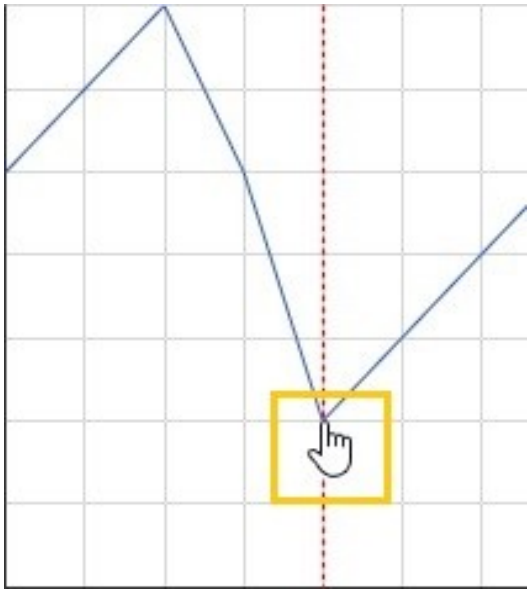
	Date/Time	Value
1	1/30/2017 14:49:18	30.00
2	1/30/2017 14:49:48	40.00
3	1/30/2017 14:50:18	50.00
4	1/30/2017 14:50:48	60.00
5	1/30/2017 14:51:18	70.00
6	1/30/2017 14:51:48	50.00
7	1/30/2017 14:52:19	20.00
8	1/30/2017 14:52:49	30.00
9	1/30/2017 14:53:19	40.00
10	1/30/2017 14:53:49	50.00
11	1/30/2017 14:54:19	60.00
12	1/30/2017 14:54:49	70.00
13	1/30/2017 14:55:19	50.00
14	1/30/2017 14:55:49	20.00
15	1/30/2017 14:56:19	30.00
16	1/30/2017 14:56:50	40.00
17	1/30/2017 14:57:20	50.00
18	1/30/2017 14:57:50	60.00
19	1/30/2017 14:58:20	70.00
20	1/30/2017 14:58:50	50.00
21	1/30/2017 14:59:20	20.00
22	1/30/2017 14:59:50	30.00
23	1/30/2017 15:00:20	40.00
24	1/30/2017 15:00:50	50.00

The viewer provides both the **data tables** and a **graphical representation**.

Double-click in the graphical representation to modify the layout of the graph. The modified layout will be saved for future use.

Zoom:

You are able to zoom in and out with the mouse wheel.



Move the cursor:

Move the mouse and the mouse cursor will change into a hand. Then click the left mouse button at the desired position and the cursor will jump near to the mouse position.

or:

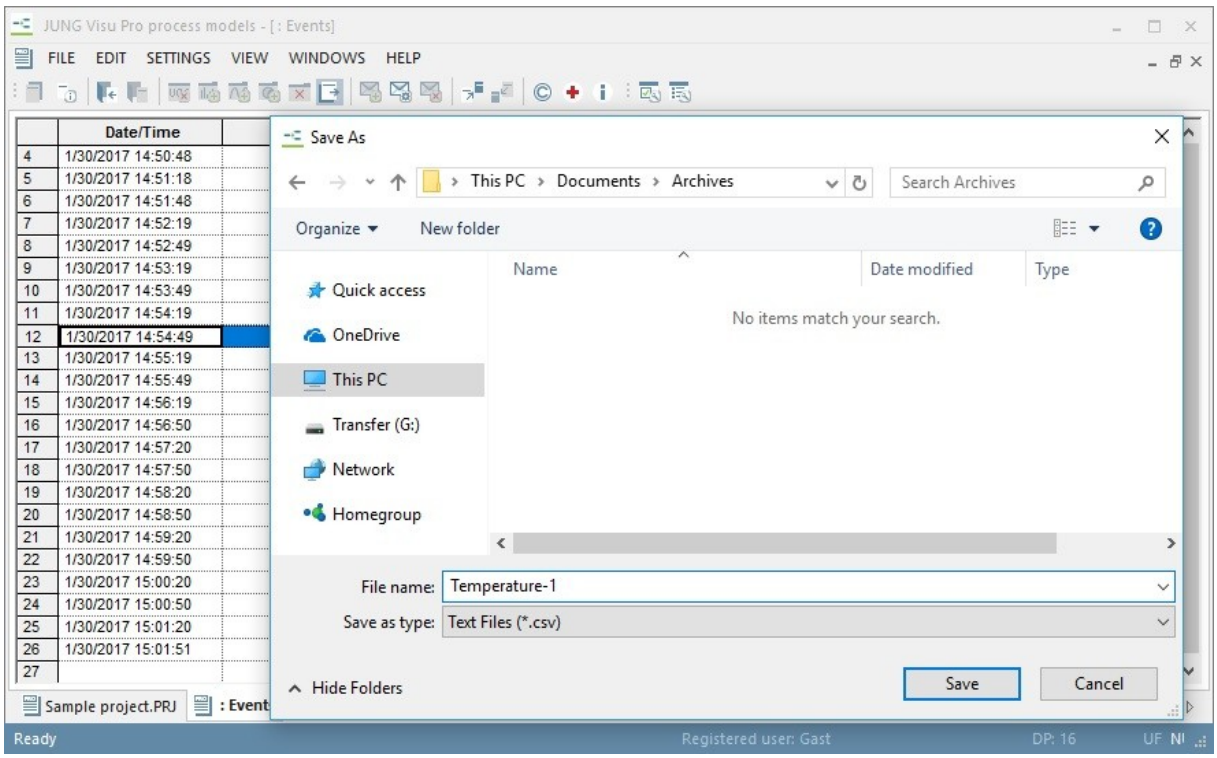
Move display rectangle:

The visible part of the display area can be moved horizontally. Left-click on the graph, hold down the mouse button and move the mouse.

7.5 Export to CSV File

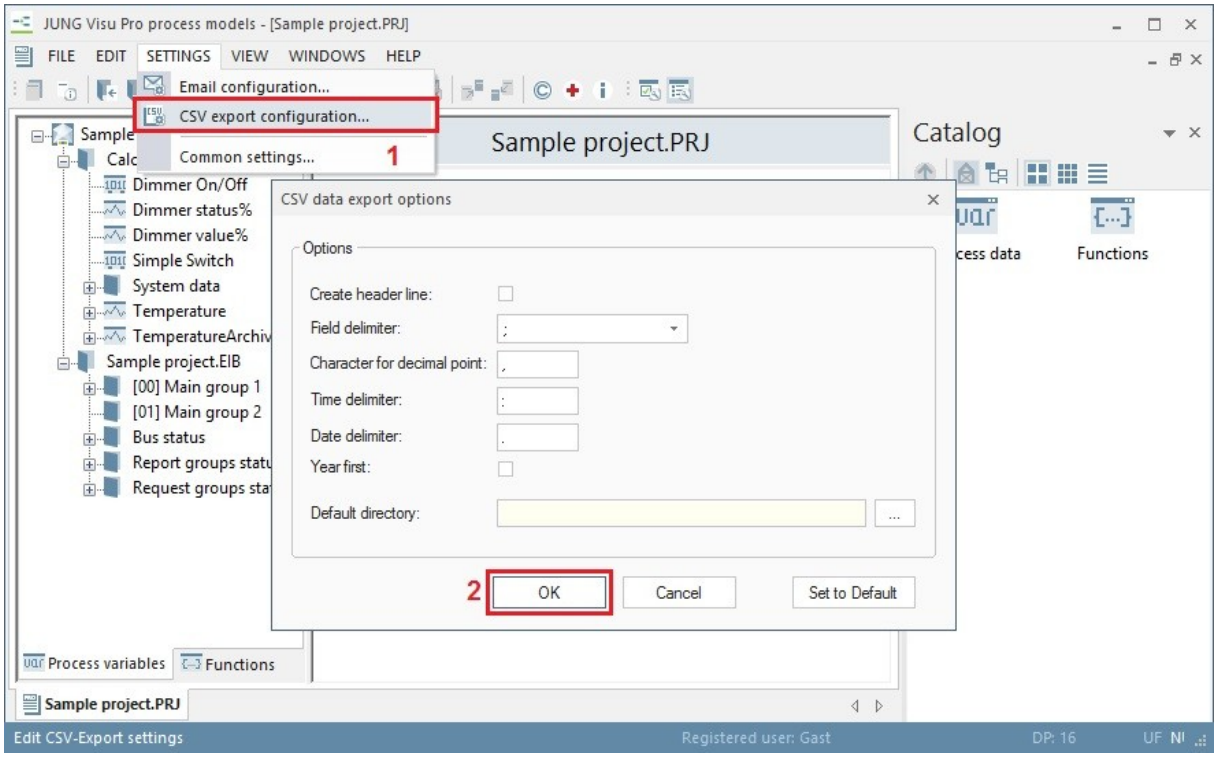
The archived values can be exported to a CSV file.

Select a range of values in the data table. Then choose the menu item ***Edit*** and click ***Export to CSV file*** or click the button in the toolbar. A file dialog opens where you can save a CSV file (*.***csv**) with the exported values. Choose the data type ***Text files***.



7.6 Configuration of CSV Export

Choose the menu item ***Settings*** and click ***CSV export configuration***. The dialog ***CSV data export options opens***.



Create header line: Specifies whether a CSV file should contain a header line or not.

Field delimiter: Selection of the CSV field delimiter (; , TAB). Default: ;

Character for decimal point: Enter the character you want to be used for the decimal point. Default: ,

Time delimiter: Enter the character you want to be used as a separator between the digits in the time specification.

Date delimiter: Enter the character you want to be used as a separator between the digits in the date specification.

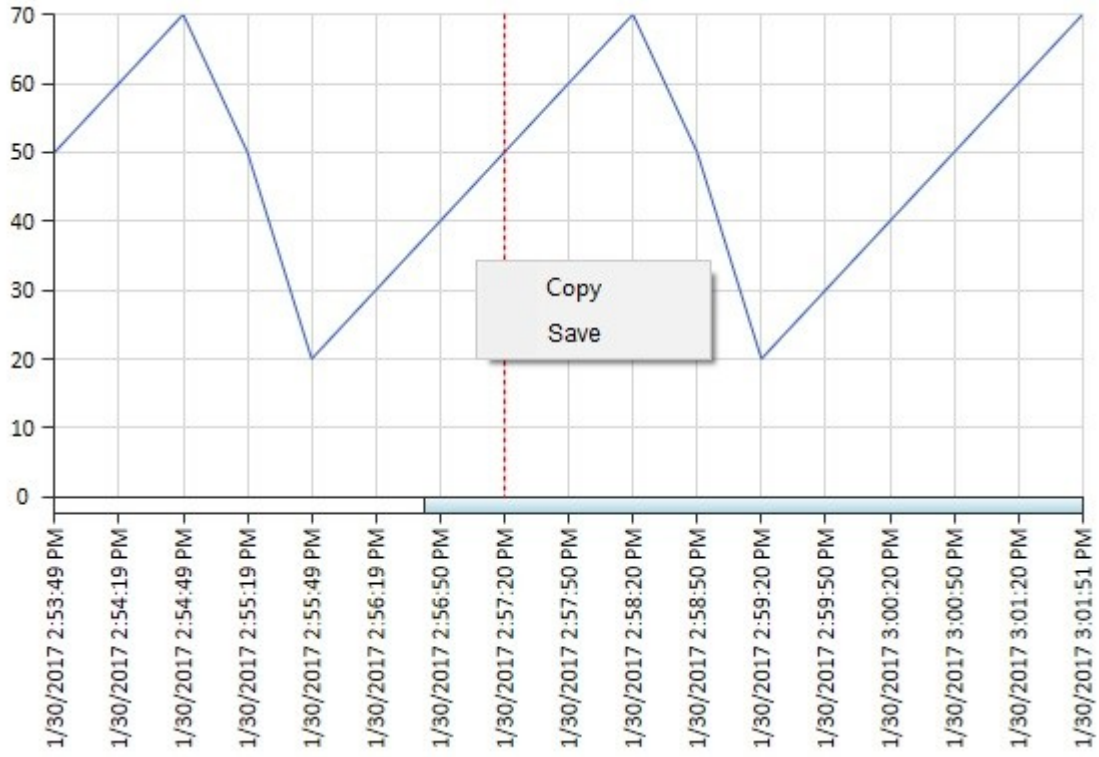
Year first: Check this option if you want the year to appear first in the data specification.

Default directory: Selection of the directory for the CSV file export.

7.7 Graph Export

The graph can be saved as BMP/JPG file or copied to the clipboard. Right-click the graph to open the ***contextmenu***.

Save graphic to file: Choose ***Save…*** In the following dialog choose the destination to which you want to save the graphic.



Copy: The graphic will be available to other programs via the clipboard.

8 Functions

The running process model is an active controller of the connected technical processes and supplements the existing installation with additional functions.

These functions include logical functions (AND, OR, XOR), status objects, forwarding, mathematical functions, time functions, counting functions (i.e. for consumption counters), sequences, scenes (i.e. light scenes), and guard functions (to secure a certain operating condition).

The evaluation of the process model runs in cycles:

a) **Fetch**
For all functions copy variables to input parameters.

b) **Execute**
For all functions evaluate output parameters.

c) **Update**
For all functions copy output parameters to the variables and if configured send values to the process interface.

d) Continue with step a)

In particular the order of the function's evaluation is not significant, since during one cycle the functions are evaluated independently.

As a precondition to the evaluation of a function, all input variables must have values. If a variable does not have a value (i. e. the value has not been read from the process), functions are not evaluated, if they use the variable as an input.

Please note that there have been considerable changes in the starting behavior, the scripts and other functions.

If you are updating from a previous version please check your project!

8.1 Starting Behavior

When the process model starts, each process variable is initialized according to its settings: either the value is initialized with a start value, a saved value or it is marked 'undefined'.

A function can only be evaluated, when all input variables have valid values and none of them is marked as 'undefined'. In earlier versions, a function did not only require valid values, but also values that were 'new', either read from the process or calculated based on 'new' values. The idea was to avoid undesired effects based on out-dated values.

Now we have learned from practical experience that it can be important to evaluate functions even when based on saved and perhaps out-dated values, in particular when self-defined process variables are used as set point variables.

While starting, the process model will now evaluate all functions with valid values, whether they are out-dated or not, but there is an option ***Starting behavior*** to further refine the starting behavior of functions: when a function is evaluated for the first time, optionally its result on process variables may be ignored.

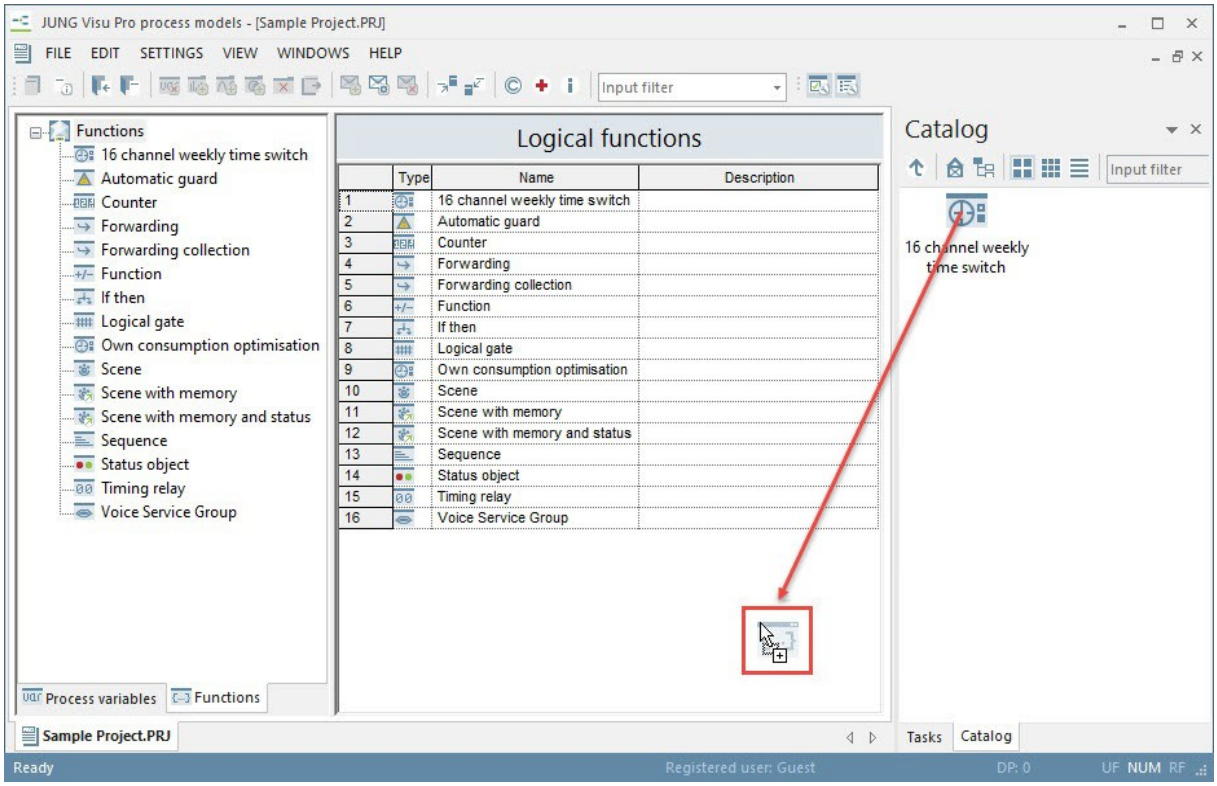
A sample with an AND – logical function shows the difference between the old and the new behavior. In both cases, the function requires valid values for the input variables. Let's say that both input variables are initialized with saved values and the function has been configured to ignore the first function's result. Let's furthermore assume that one input is a set point variable, the other read from the process.

Then the reading from the process will lead to valid function result in the next cycle, while according to the old behavior, both a reading from the process and changing the set point variable would be required for the function to execute.

8.2 Add Function

Switch in the window on the left side of the editor to the tab ***Functions***. Make sure that the root directory Functions is selected. The main window displays a grid with all controller functions configured so far.

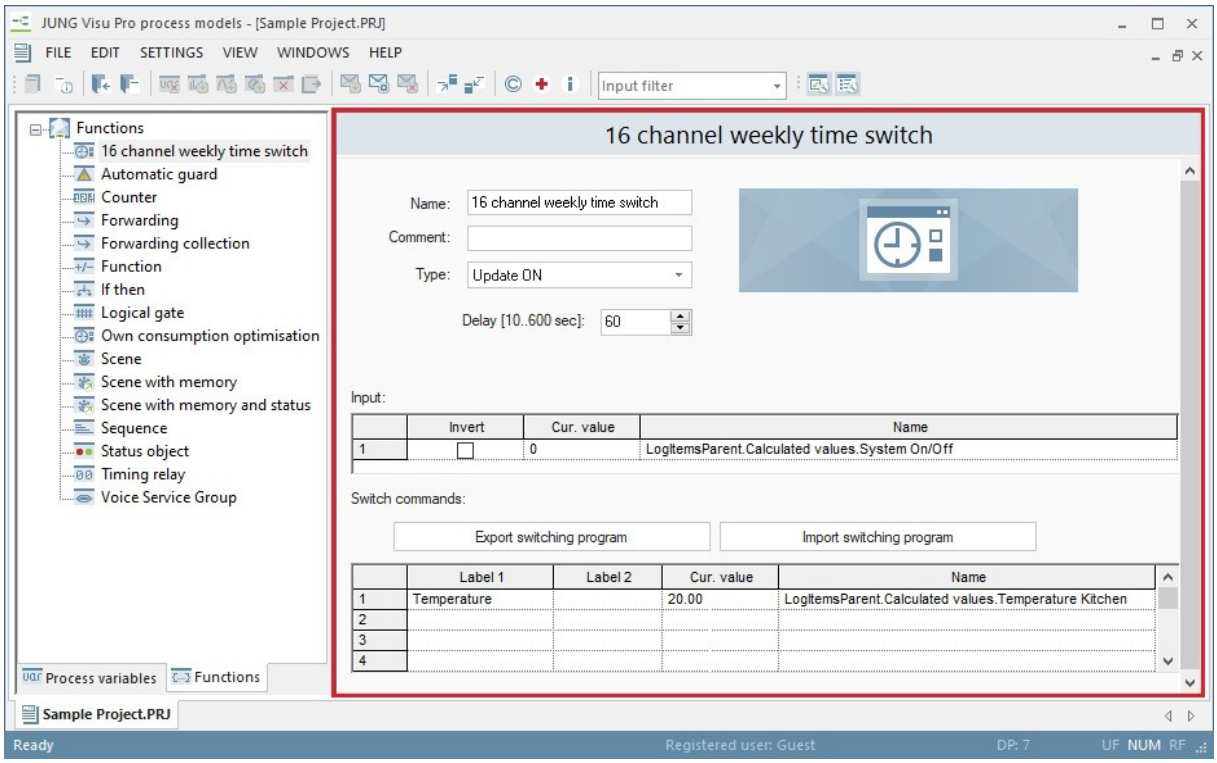
Then choose the tool window ***Catalog*** and select the category ***Functions***. Drag the desired function to the function grid in the main window.



Edit functions:

- In the function grid double-click the function symbol in the column ***Type*** or
- click the function in the structure tree in the left window of the editor.

The main window will show the configuration of the function.



Parameterize the functions:

The parameterization of functions is done by dragging process variables to the function. For this purpose, choose the tool window ***Catalog*** and select the category ***Process data***. Then drag the desired process variable to the function.

To delete a process variable from the function, select the row in which the process variable resides by clicking its number column. Then press the keyboard key ***Delete***.

8.3 Logical Gate

Function description:

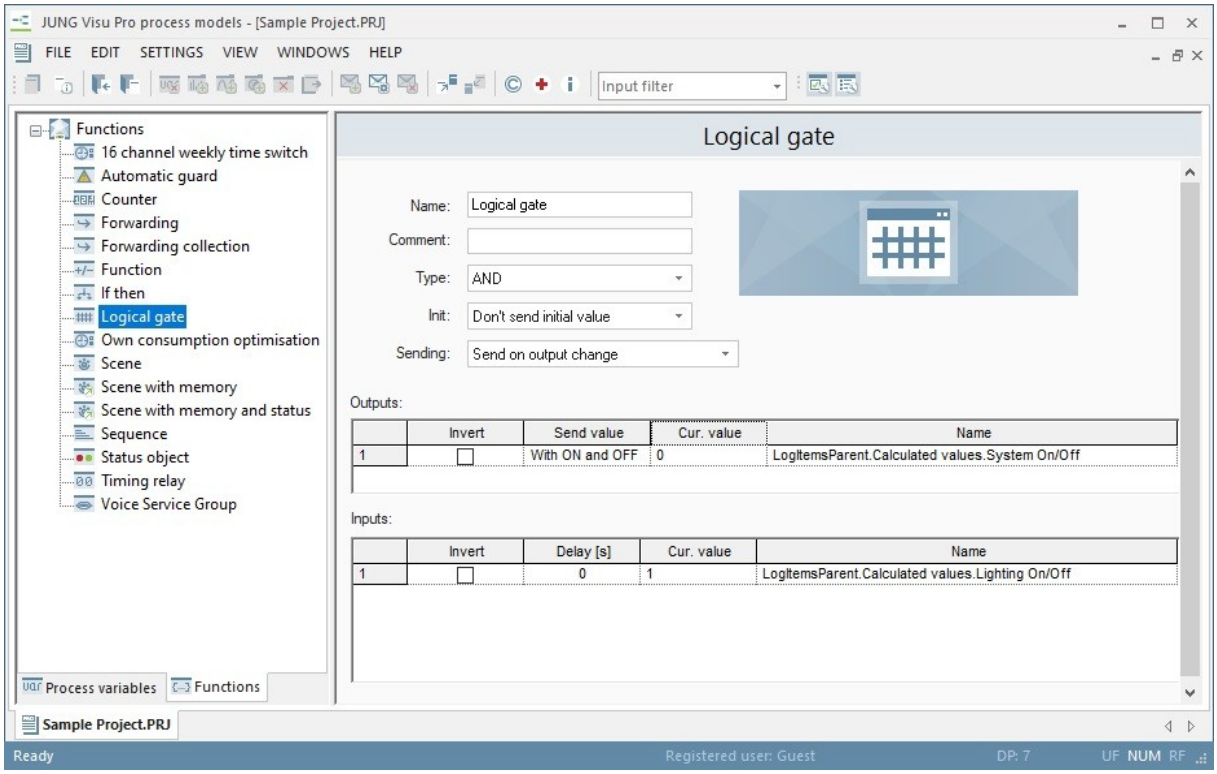
The logical gate is used to configure logical connections. It may be an AND, OR or XOR (logical exclusive or) gate.

The gate's logical function will be applied to the input variables, the result will be assigned to the output variable. Whether the value of the output variable will be sent to the process interface (i.e. the KNX group address will be sent) depends on the configuration.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Type: Select the logical function: AND, OR or XOR.

Init: Allow or disallow sending the function result when the function is evaluated the first time, i.e. after the process model has been started.

Sending: Decide when to send the function result.

On changing output: The function result will only be sent when it has been changed.

On input update: The function result will always be sent when an input has been updated, regardless of the input value.

Outputs: Process variables which will hold the function result.

Inputs: Process variables as the inputs of the function.

Settings for each output:

Invert: Decide whether the output should be inverted.

Sending: Choose a condition when to send the output value to the process interface.

Cur. value: This column displays the current value of the process variable.

Settings for each input:

Invert: Decide whether the input should be inverted.

Delay: Define a delay in seconds for the evaluation of changed input values. Changes will be ignored, if they are not stable for at least the delay period.

Cur. value: This column displays the current value of the process variable.

8.4 Mathematics

Function description:

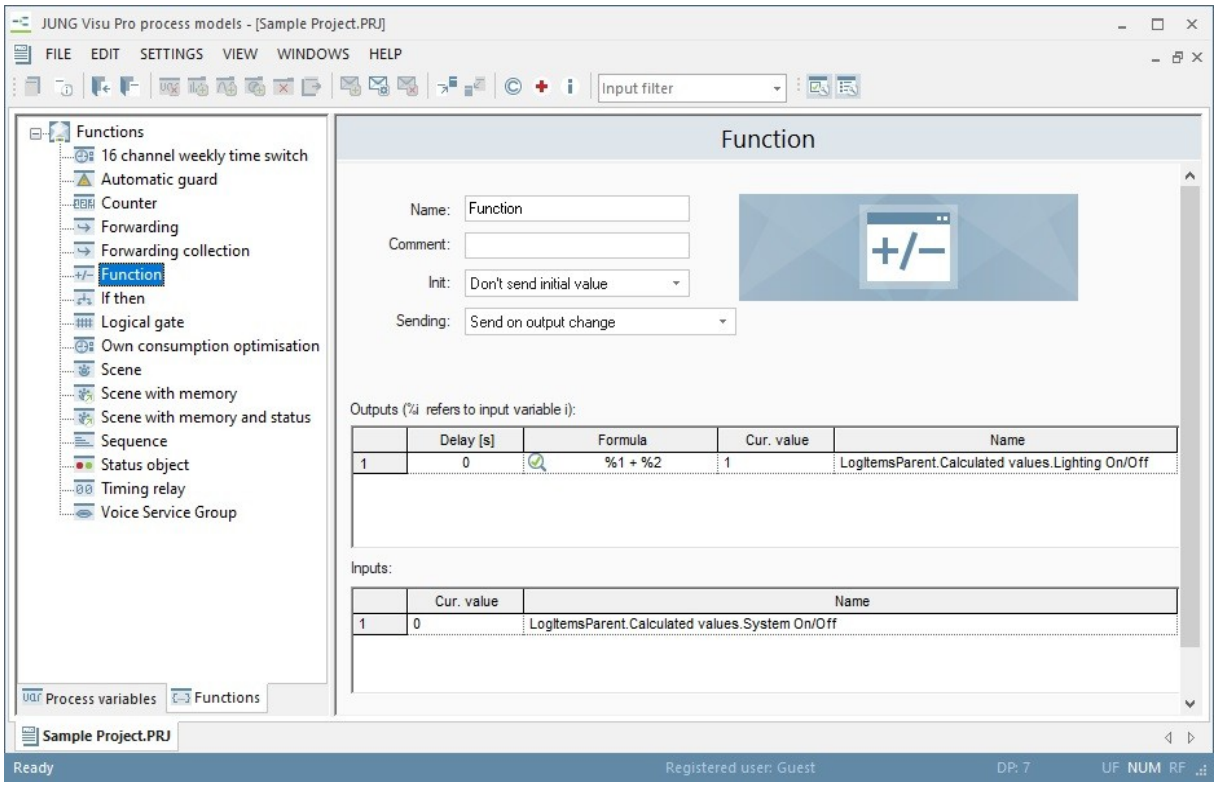
Mathematical functions are used to calculate the value of an output variable based on a formula and one or more input variables.

Whether the value of the output variable will be sent to the process interface (i.e. the KNX group address will be sent) depends on the configuration.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input variables or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Sending: Decide when to send the function result.

On changing output: The function result will only be sent when it has been changed.

On input update: The function result will always be sent when an input has been updated, regardless of the input value.

Outputs: Process variables which will hold the function result.

Inputs: Process variables as the inputs of the function.

Settings for each output:

Delay: Define a delay in seconds for the evaluation of changed input values. Changes will be ignored if they are not stable for at least the delay period.

Formula: Enter a formula to calculate the output value, with "%1", "%2" .. (without quotes) referring to the first, second, .. input variable.

Cur. value: This column displays the current value of the process variable.

Note: If any value of the input variables is unknown, the evaluation is terminated and output variables will not be updated.

8.5 Sequence

Function description:

The sequence function is a time schedule of several commands to either set or query the value of a process variable. After each command the function will delay execution until a delay period has passed.

The delay period will be specified in seconds. There are two special cases for the delay period:

- 0 seconds: No delay
- An interval, i.e. 10 – 30: The delay will be a random value in this interval.

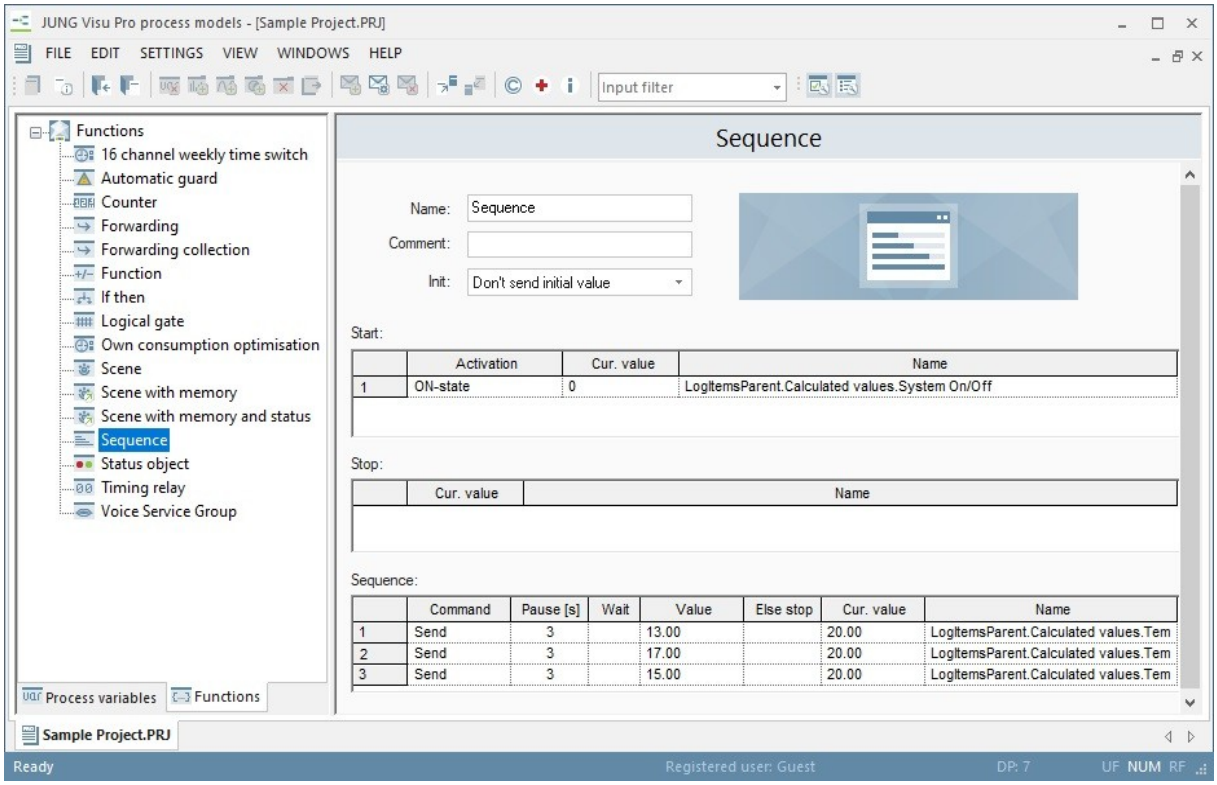
As an extension to simple sequences of read or write commands, the sequence may check the state of a variable. If the condition on the variable, given as a formula, has not been met within a certain time period or after a certain time period, the sequence may be stopped. If stopped, an error variable may be set accordingly.

A sequence is controlled by a binary process variable. It may be started when the variable is changing (activation by pulse) or may repeatedly run as long as the binary variable has a certain value.

Configuration:

Choose the catalog window and select the category ***Process data***. Then drag the process variable to the grid of input variables or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function
Comment: Enter a short description of the function.
Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Start: Binary process variable controlling the sequence.

In the column ***Activation*** select the activation method for the sequence:

- **"pulse off":** The sequence will be started when the value of the process variable changes from "on" (value 1) to "off" (value 0).
- **"pulse on":** The sequence will be started when the value of the process variable changes from "off" (value 0) to "on" (value 1).
- **"pulse on/off":** The sequence will be started when the value of the process variable is "on" (value 1) or "off" (value 0).
- **"state off":** The sequence will be repeatedly run while the process variable state is "off" (value 0).
- **"state on":** The sequence will be repeatedly run while the process variable state is "on" (value 1).

Cur. value: This column displays the current value of the process variable.

Stop: Optionally, a binary process variable may be dragged to this field. The value of the used variable is set from 0 to 1, if a defined condition is fulfilled. A reset must be done outside the sequence. Either manually or from the process model. The variable must be reset directly.

Sequence: Commands on one or more process variables.

For each row in the sequence:

- **Command:** In this column you have to define whether it is a send command or a check command.
  Send commands: Send commands are sent from the sequence. Sending may be either sending a query telegram or a write telegram. When sending a write telegram, the column ***Value*** contains the value that has to be written.
  Check commands: For check commands, the condition to be checked must be specified in the ***Value*** column. The condition variable is always defined with %1 and refers to the process variable of the row. In case of checking, the column ***Value*** contains a formula. Checking is successful if the formula returns true.
- **Pause:** Here you specify a period in which the sequence pauses before the command is executed. When sending, the command will be delayed for the specified number of seconds. Instead of the number of seconds an interval of seconds, i.e. 10 – 20 can be specified. In this case the sequence will use a random number of seconds in this interval. When delay is 0, the sequence will immediately execute the command. Checking expects that the formula in the column ***Value*** will return true within the delay time interval.

- **Wait:** This column applies to checking only. If "waiting" is turned on, the sequence will wait until the delay time has passed even when checking has already been successful before. Otherwise the sequence will continue as soon as the formula in the column ***Value*** returns true even if the time period has not passed completely. The attribute **Wait** causes the system to wait for the period defined in ***Pause*** after fulfilling the condition. If the attribute is not activated, a fulfilled condition is evaluated immediately. If the conditions are not fulfilled, the system waits for the pause time, during which the condition can still be fulfilled.

- **Value:** In this column the executing value is to be determined. When sending a write telegram this is the value to write. When checking, this field comprises a formula. Refer to the process variable in the row with "%1" (without quotes). In case of a formula you may double-click the formula symbol to open a dialog to check the formula.

- **Else stop:** This column applies to check commands only and causes the stop behavior of the sequence (defined by the activation) if the condition is not fulfilled. If a check command is enabled, the sequence will be stopped if checking has not been successful within the time period. The process variable for the stop state – if specified – will be set to 1 and this can indicate, e.g. in the visualization, that the sequence has not run to the end (can also trigger an alarm, for example). The process variable can be reset manually or from the process model. The process variable for the stop state does *not* stop the sequence, it is activated as soon as the sequence stops or restarts due to an unfulfilled condition.

- **Cur. value:** This column displays the current value of the process variable.

Note: *The stop variable could be used to indicate a problem, i.e. could be an alarm variable.*

Note:

- ***Pulse-controlled stop behavior:*** *If check conditions are defined, the sequence is stopped after checking an unfulfilled condition.*
  State-controlled stop behavior: *If check conditions are defined, the sequence is started from the beginning after checking an unfulfilled condition, so the sequence runs infinitely and is to be stopped via an inverted start state. A sequence controlled by a state will automatically restart with the first command. However, you can check the error variable at the beginning of the sequence to avoid execution of the following commands.*

- *Checking of variables can be useful to disable/enable a sequence. For example a variable could be checked which is set by the calendar application.*

8.6 Status Object

Function description:

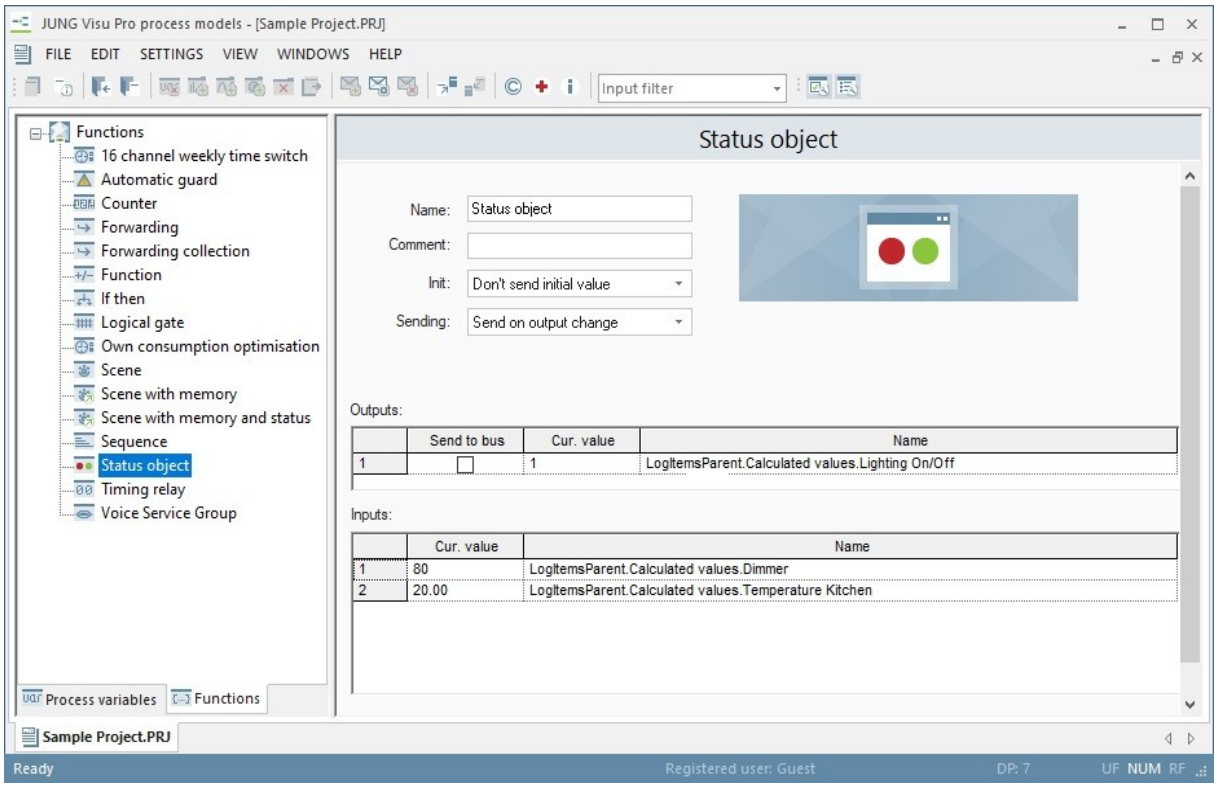
The status object updates the output process variable with one or more input process variables. The value of the output variable is always the value of the input variable last changed.

Whether the output variable value will be sent to the process interface (i.e. the KNX group address will be sent) depends on the configuration.

Configuration:

Choose the catalog window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Sending: Decide when to send the function result.

On changing output: The function result will only be sent when it has been changed.

On input update: The function result will always be sent when an input has been updated, regardless of the input value.

Output: Process variable which will hold the function result.

Inputs: Process variables as the inputs of the function.

Cur. value: This column displays the current value of the process variable.

Output settings:

Sending: Decide whether the output value will be sent to the process interface.

8.7 Scene

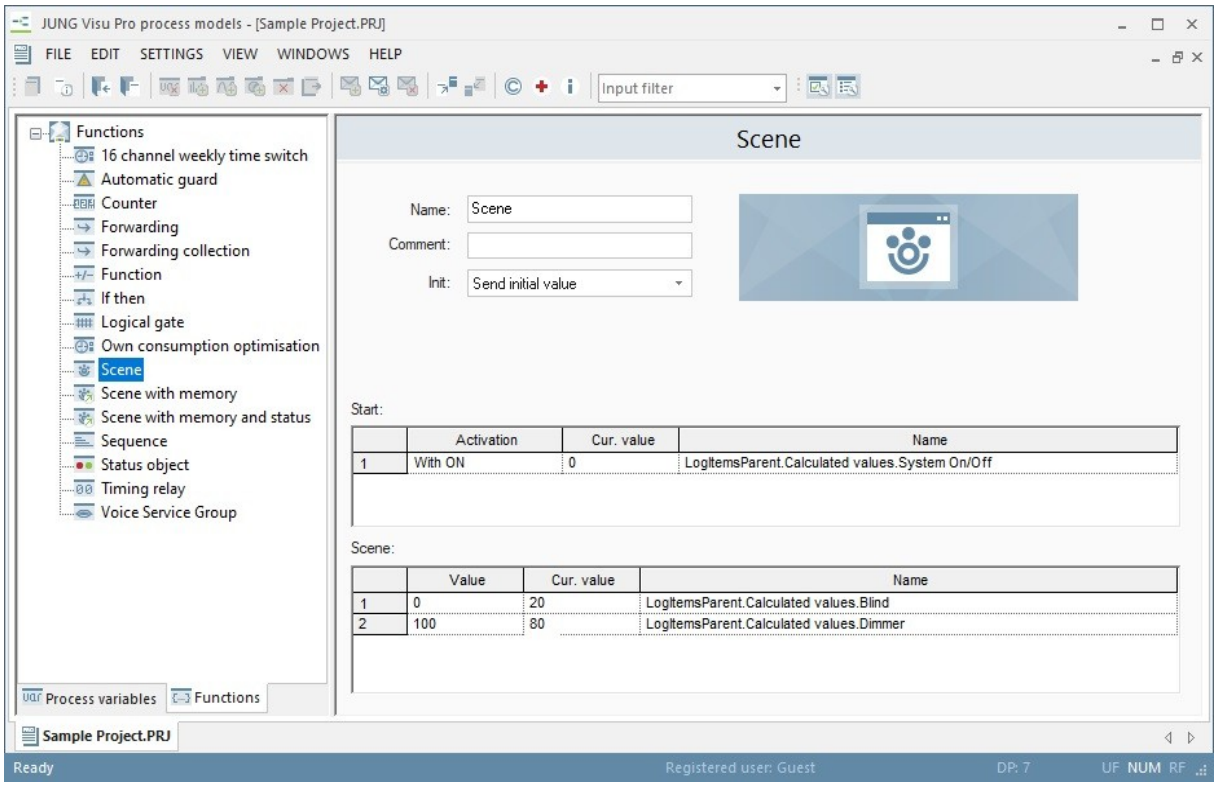
Function description:

The scene function is used to send, write or read commands to several process variables at the same time (in practice one immediately after the other). It is controlled by a binary process variable.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input variables or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Start: Binary process variable controlling the sequence.

In the column ***Activation*** select the activation method for the scene function:

- **"state off":** The scene will be started when the value of the variable changes from "on" (value 1) to "off" (value 0).

- **"state on":** The scene will be started when the value of the variable changes from "off" (value 0) to "on" (value 1).

- **"state on or off":** The scene will be started when the value of the variable changes.

Scene: Commands on one or more process variables.

For each row in the scene:

Value: Either the value to be written or a query command.

Binary process variable: Select command from the combo box.
Analog process variable: Directly enter the value or enter "read" (abbreviation "re"). When "read" has been entered, the scene function will send a read telegram instead of a write telegram.

Cur. value: This column displays the current value of the process variable.

8.8 Scene with Memory

Function description:

Additionally to the functionality of *Scene*, *Scene with memory* allows the user to save the **current** values of all contained process variables in a scene memory. The saved values will be sent to the process when the user calls the scene. The writing of current values into the scene memory can be controlled by one or more process variables. The table ***Save*** contains these control variables.

This function allows to predefine a scene, which can be dynamically changed by the user:

- Create the 'scene with memory' function in the process model.

- As in the standard scene, specify the variables for output and scene call.

- Additionally, specify one or more variables to control the saving of the current state of the output variables in the scene memory (table *Save*). In this table, you can insert a calculated process variable which is visible in the visualization only, or you can insert a process variable which is connected with a 'real' switch.

- The user may set the desired state of the scene's output variables in his installation (e.g. lights on/off, dimmer values, ...) and press the switch which is connected to one of the saving control variables. The corresponding control variable will change its state and thus start the transfer of the current values into the scene memory.

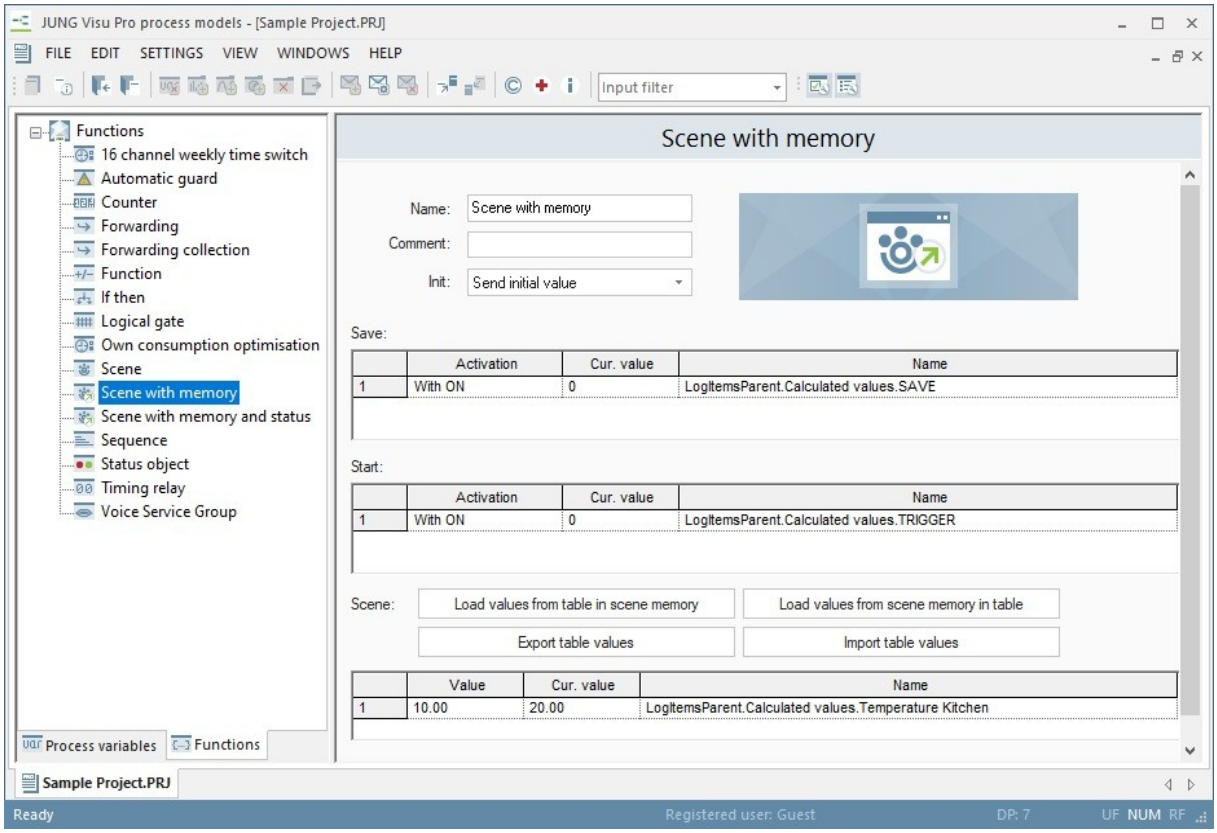
- By the next call of the scene the new values will be read from the scene memory and sent to the process.

The content of the scene memory can be loaded into the table ***Scene*** by clicking the button ***Load values from scene memory in table***. In the table the values can be changed and then saved by clicking the button ***Load values from table in scene memory***.

Configuration:

Choose the catalog window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Save: Table with one or more binary process variables to control the transfer of current values into the scene memory.

Start: Table with one or more binary process variables to call the scene.

In the column ***Activation*** select the activation method for the scene function:

- **"state off":** The scene will be started when the value of the variable changes from "on" (value 1) to "off" (value 0).
- **"state on":** The scene will be started when the value of the variable changes from "off" (value 0) to "on" (value 1).
- **"state on or off":** The scene will be started when the value of the variable changes.

Scene: All process variables of the scene and the corresponding output values.

Load values from table in scene memory: Writes the output values from the table in the scene memory.

Load values from scene memory in table: Loads the output values from the scene memory in the table.

Export table values / Import table values: To save the current values, the scene can be exported in the form of a cfg file. This file can be imported again by clicking *Import table values*.

Note: The export only makes sense if scene values are assigned.

For each row in the scene:

Value: Either the value to be written or a query command.

Binary process variable: Select command from the combo box.
Analog process variable: Directly enter the value or enter "read" (abbreviation "re"). When "read" has been entered, the scene function will send a read telegram instead of a write telegram.

Cur. value: This column displays the current value of the process variable.

8.9 Scene with Memory and Status

Function description:

The ***scene with memory and status*** is a function for use with the same named JUNG-specific control element that can be operated and adjusted in the HTML interface of the server.

The ***scene with memory and status*** allows the user to save the **current** values of all contained process variables in a scene memory. The saved values will be sent to the process when the user calls the scene. The writing of current values into the scene memory can be controlled by one or more process variables. The table ***Save*** contains these control variables.

The values of a scene will not updating themselves right after send, like it is within the ***scene with memory***. Instead of, the ***scene with memory and status*** works with a second value at the same channel (feedback object). The scene value only gets a change by an incoming update from the feedback object. This also needs to be dragged into the scene column.

In addition, there is an option to insert display names, which are used in the server to display the scenes channel.

Configuration:

Choose the catalog window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

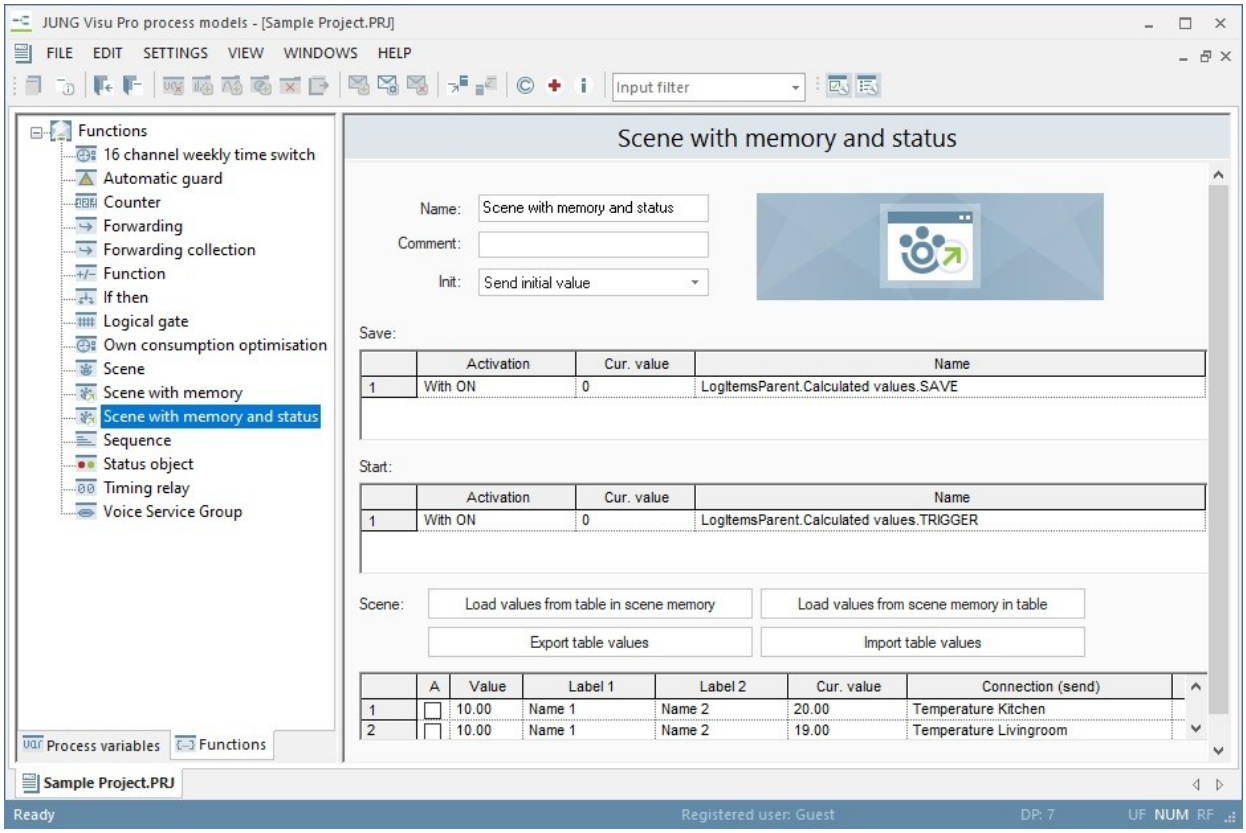
To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.

Start behavior:

- Evaluate initial value
- Do not evaluate initial value

Meaning: The first time an event is received, this parameter determines whether the first received value can trigger the scene or not. This parameter also determines whether the first value received for saving the scene already performs the action.

Note: The memory scene does not respond to the change in the value of an input variable (OFF to ON), but to the value of the event message itself. Consecutive messages with the same value trigger the memory scene also, that means that the previous value of the input variable is not significant. This also applies to the input variable to store the scene.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Save: Table with one or more binary process variables to control the transfer of current values into the scene memory.

Start: Table with one or more binary process variables to call the scene.

In the column ***Activation*** select the activation method for the scene function:

- **"state off":** The scene will be started when the value of the variable changes from "on" (value 1) to "off" (value 0).
- **"state on":** The scene will be started when the value of the variable changes from "off" (value 0) to "on" (value 1).
- **"state on or off":** The scene will be started when the value of the variable changes.

Scene: All process variables of the scene and the corresponding output values.

Load values from table in scene memory: Writes the output values from the table in the scene memory.

Load values from scene memory in table: Loads the output values from the scene memory in the table.

Export table values / Import table values: To save the current values, the scene can be exported in the form of a cfg file. This file can be imported again by clicking *Import table values*.

Note: The export only makes sense if scene values are assigned.

For each row in the scene:

Value: Either the value to be written or a query command.

Binary process variable: Select command from the combo box.
Analog process variable: Directly enter the value or enter "read" (abbreviation "re"). When "read" has been entered, the scene function will send a read telegram instead of a write telegram.

Cur. value: This column displays the current value of the process variable.

8.10 Forwarding

Function description:

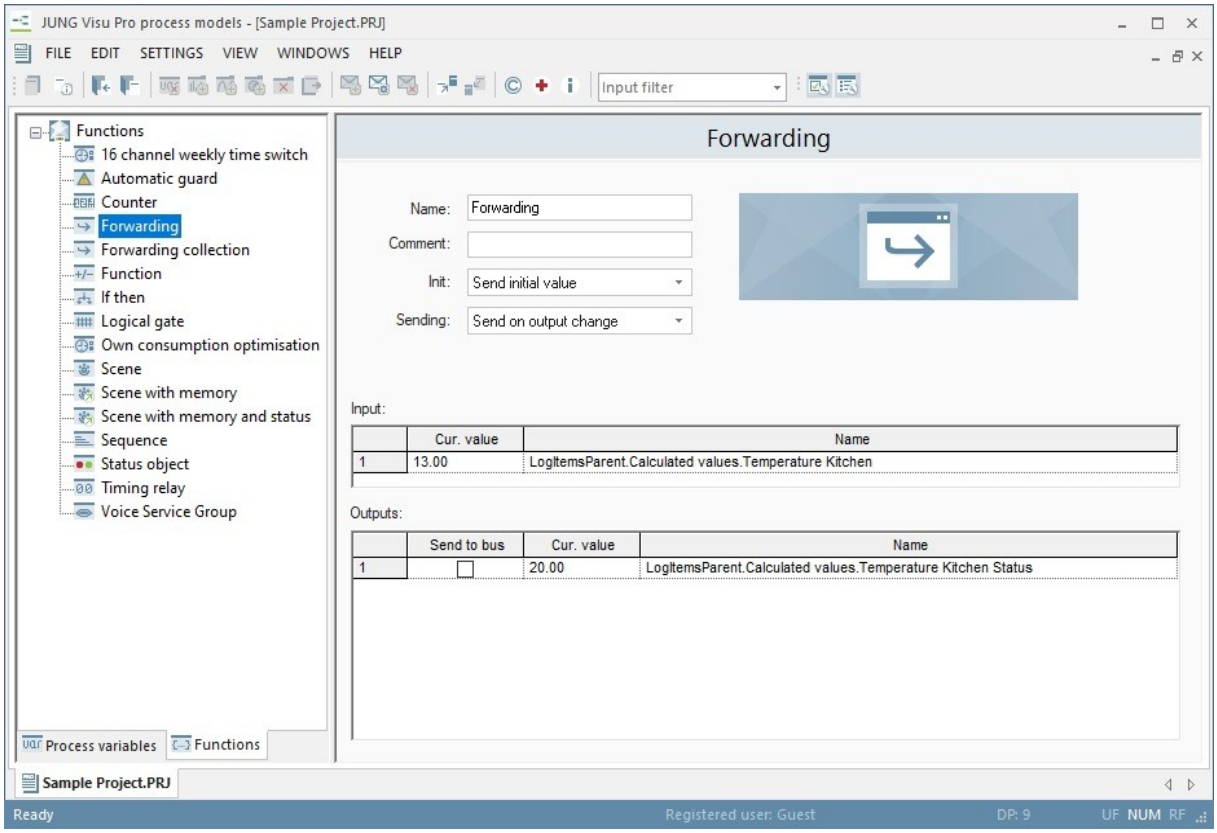
The function *forwarding* connects one input process variable to one or more output variables. The value of the input variable will be simply copied to the output variables. Though no computation is implied, this function can be very useful to implement gateways from one technical subsystem (for instance connected with OPC) to another (for instance connected with KNX).

Whether the value of an output variable will be sent to the process interface (i.e. the KNX group address will be sent) depends on the configuration.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Input: Process variable.

Sending: Decide when to send the function result.

On changing output: The function result will only be sent when it has been changed.

On input update: The function result will always be sent when an input has been updated, regardless of the input value.

Outputs: Process variables which will hold the function result.

Cur. value: This column displays the current value of the process variable.

Settings for each output:

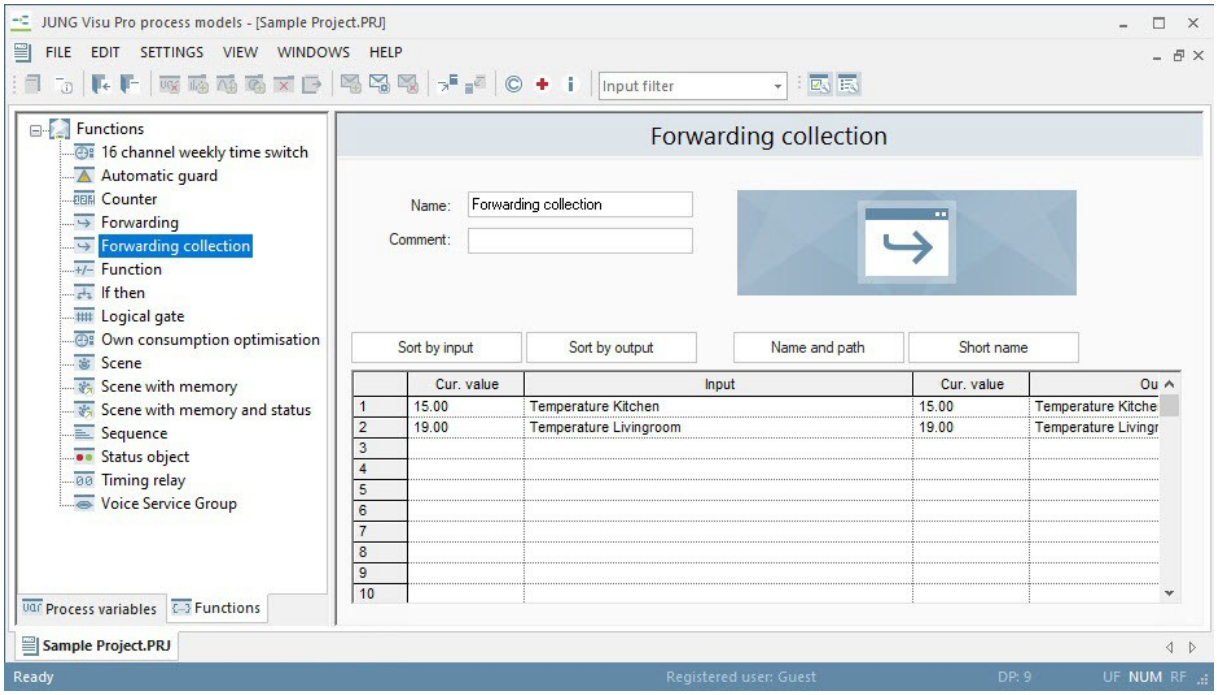
Sending: Decide whether the output value will be sent to the process interface.

8.11 Forwarding Collection

Function description:

The forwarding collection is used to forward values of one variable (input variable) to an output variable. The value of the input variable will be simply copied to the output variable.

The forwarding collection only reacts to events, so the output event will be only updated, if the input value is set to a new value. Every entry in the forwarding collection list contains out of an input and an output variable.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Output variable: Field for the target variable of the forwarding collection. The given variable will receive the values of the input variable.

Input variable: Field for the source variable of the forwarding collection.

Sort by output: Sorts all entries in alphabetical order by the name of the output variables. The path, if shown will be considered.

Sort by input: Sorts all entries in alphabetical order by the name of the input variables. The path, if shown will be considered.

Name and path: Displays the names of all variables with its path in the table.
A path contains the project, as well as the main and middle groups.

Short name: Only displays the name of the variable in the table.

8.12 If Then

Function description:

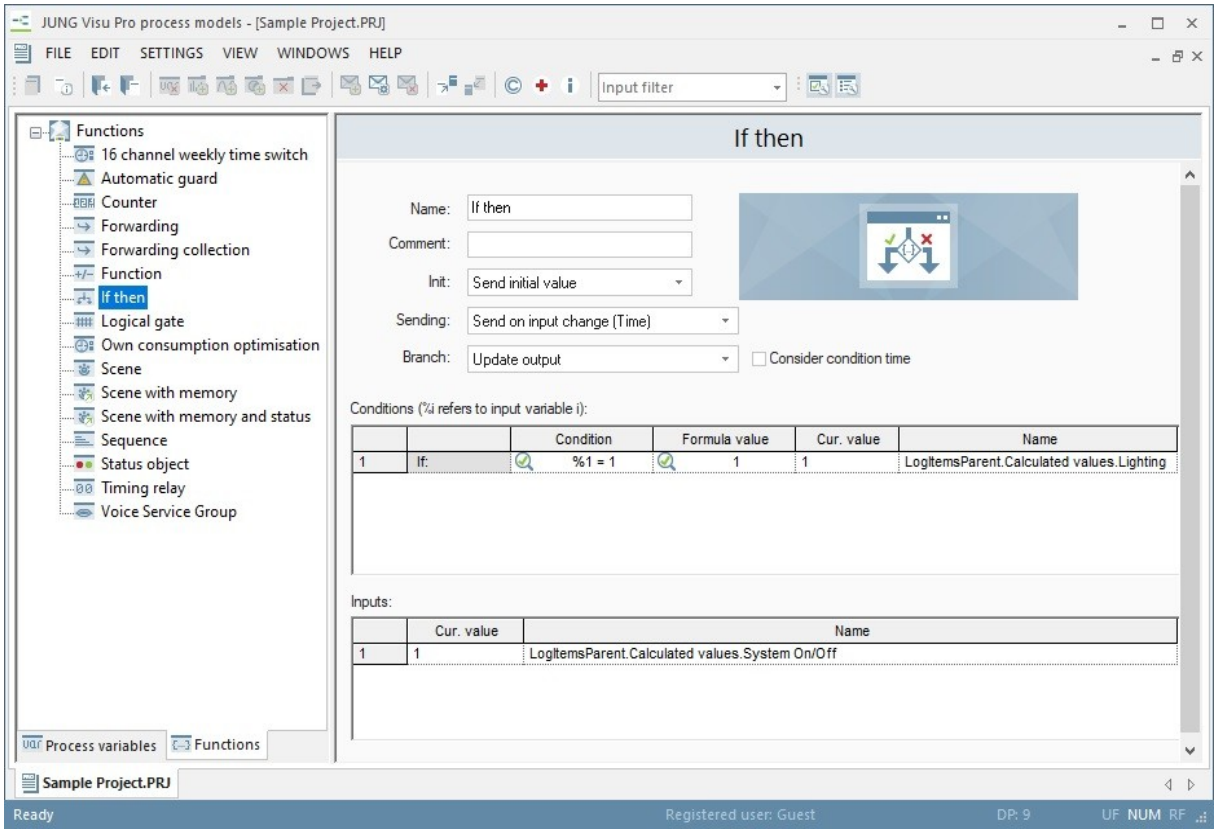
The if-then function is used to calculate the values of process variables depending on other process variables and conditions.

It can be used, for instance, for the configuration of a priority control: The value of a switch will only be sent to an actor if this connection has been allowed. The allowance may depend on another binary process variable.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input variables or the condition grid. If dragged to the inputs grid, the process variable can be used in the columns ***Condition*** and ***Value***. If dragged to the condition grid, a new condition row will be created and the process variable is used as an output variable.

To delete a condition from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated the first time, i.e. after the process model has been started.

Sending: Decide when to send the function result.

On changing output: The function result will only be sent when it has been changed.

On input update: The function result will always be sent when an input has been updated, regardless of the input value.

Branch:

For an output variable there can be two reasons to calculate a new value:

- Due to the evaluation of conditions, a different if-else branch (row) will be applied.
- Still in the same if-else branch the value changes.

Active branch: Always update output variables from the ***Value*** column of the applicable if-else branch (row).

Passive branch: If for an output variable a new if-else branch will be applied (a different condition will be valid), do not update the output variable value. Next time the value of the row is recalculated due to a change of input variables, the output variable will be updated.

This option is important if conditions are used to enable/disable a certain behavior but enabling/disabling is not supposed to send a value.

Conditions:

The condition grid comprises a sequence of if-else branches. When evaluating the conditions, the function will work down the rows, starting with the first row, and evaluate the ***Condition*** column until it finds a row with a valid condition. Then it evaluates the formula in the column ***Value***, assigns the value to the output variable in the same row, then stops.

For conditions and values a left click on the formula symbol will open a dialog to check formulas. Refer to input variables with "%1", "%2",.. (without quotes) to refer to the input variable at the first, second, .. row in the input variable grid.

The editor will check for erroneous entries and mark them red.

Inputs: The process variables used in ***Condition*** and ***Value*** formulas.

Note: If any value of the input variables is unknown, the evaluation is terminated and output variables will not be updated.

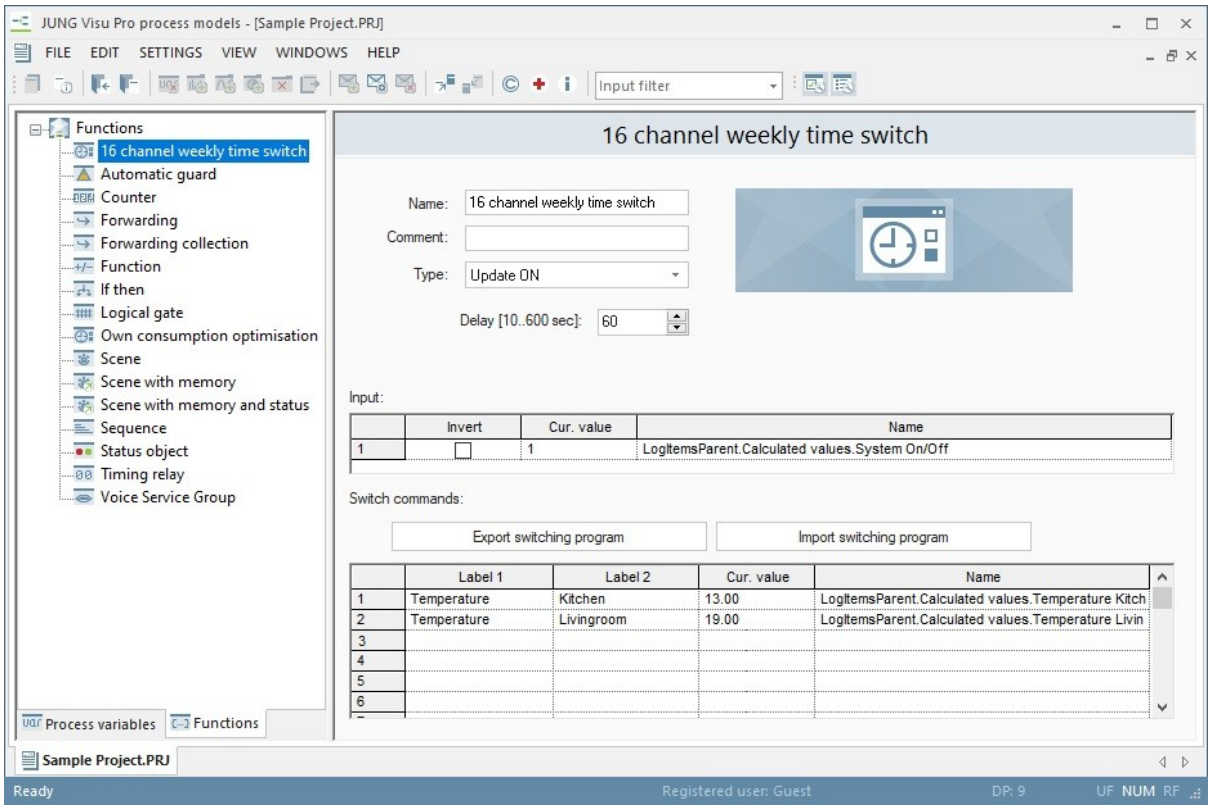
8.13 16 Channel Weekly Time Switch

Function settings:

The 16 channel weekly time switch is a function for the programming of the same named JUNG-specific control, which is only configurable in the HMTL-surface. The time the weekly time switch uses is based on the system time.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.
To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Name: Enter the name of the function.

Comment: Enter a short description of the function.

Type: Function selection: Time adjustment On or Off.

Delay: Defines a delay in minutes. The delay takes effect after the start of the process model. It indicates how long the activation of the 16 channel weekly time switch takes after the start of the process model.

Input: Inserts binary process variables for the activation of the 16 channel weekly time switch.

Switch commands: Inserts variables for the time-controlled scenes of the 16 channel weekly time switch. Each 16 channel weekly time switch function can only contain a **maximum of 16 variables**.

Export switching program / Import switching program: To save the current switching times, the switching program can be exported in the form of a cfg file. This file can be imported again by clicking *Import switching program*.

Note: The export only makes sense if switching times are assigned.

8.14 Counter

Function description:

The counter function can be used to implement counters for pulse or operating hours, based on "on" and "off" messages from binary counters.

Counter for pulses:
The number of messages is counted.

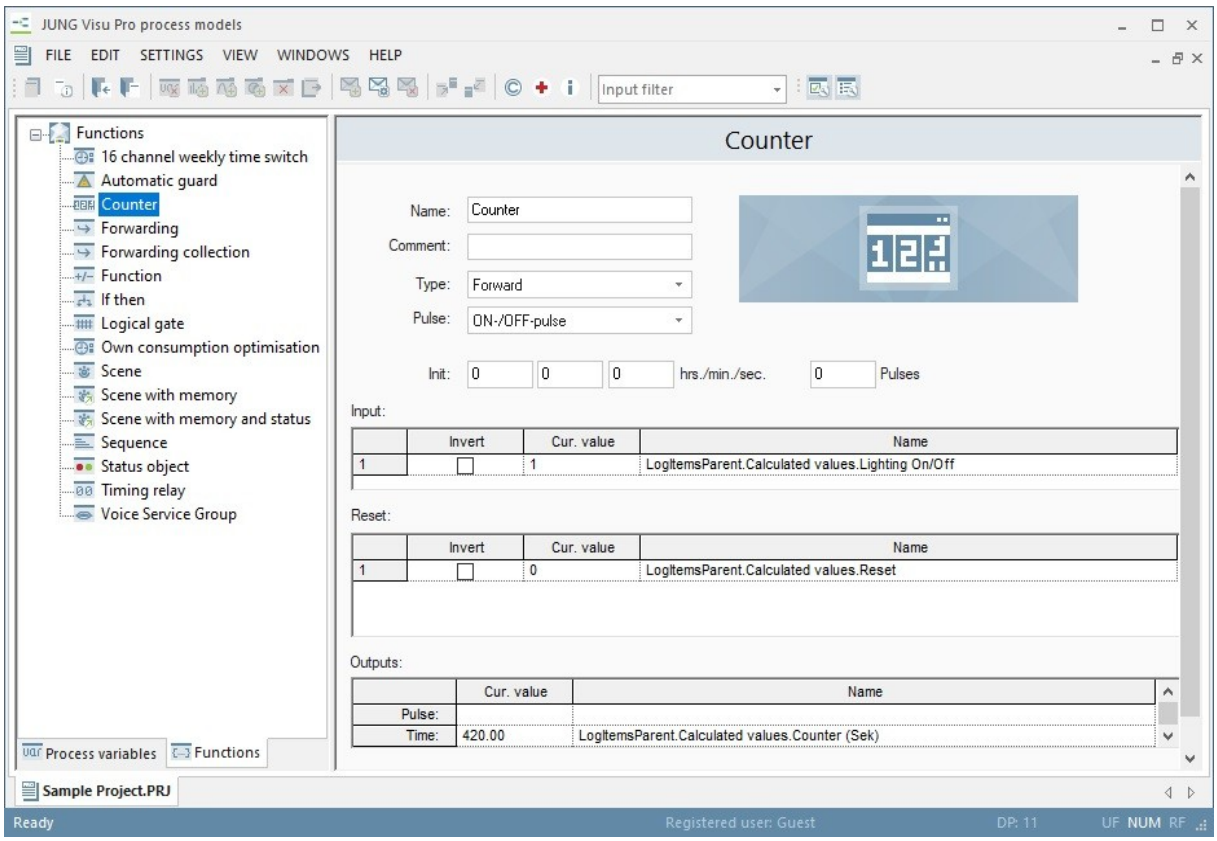
Counter for operating hours:
The time between "on" and "off" messages will be counted. The hours, minutes and seconds can be assigned to separate variables.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.

For time registration, the row ***Time*** must be connected to an analog process variable. Then additional rows will allow to connect separate variables for the hours, minutes and seconds.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Type: Select whether the counter should decrement or increment.

Pulse: Select whether to evaluate only "on" messages, only "off" messages or both.

Init: Initialize the function with a starting time (hours/minutes/seconds) and a start value for the pulse counter.

Input: The binary input variable.

Reset: Optionally a reset variable can be assigned. When the value of the reset variable is changing from "off" (value 0) to "on" (value 1) the counter will be reset to the starting values. If the reset variable is to be inverted, reset will be performed when the value is changing from "on" to "off".

Outputs:

Process variables, which will hold the results.

Pulses:	The number of pulses counted.
Time:	The total time in seconds.
Hours:	The hours fraction of the counted time.
Minutes:	The minutes fraction of the total time.
Seconds:	The seconds fraction of the total time.

8.15 Timing Relay

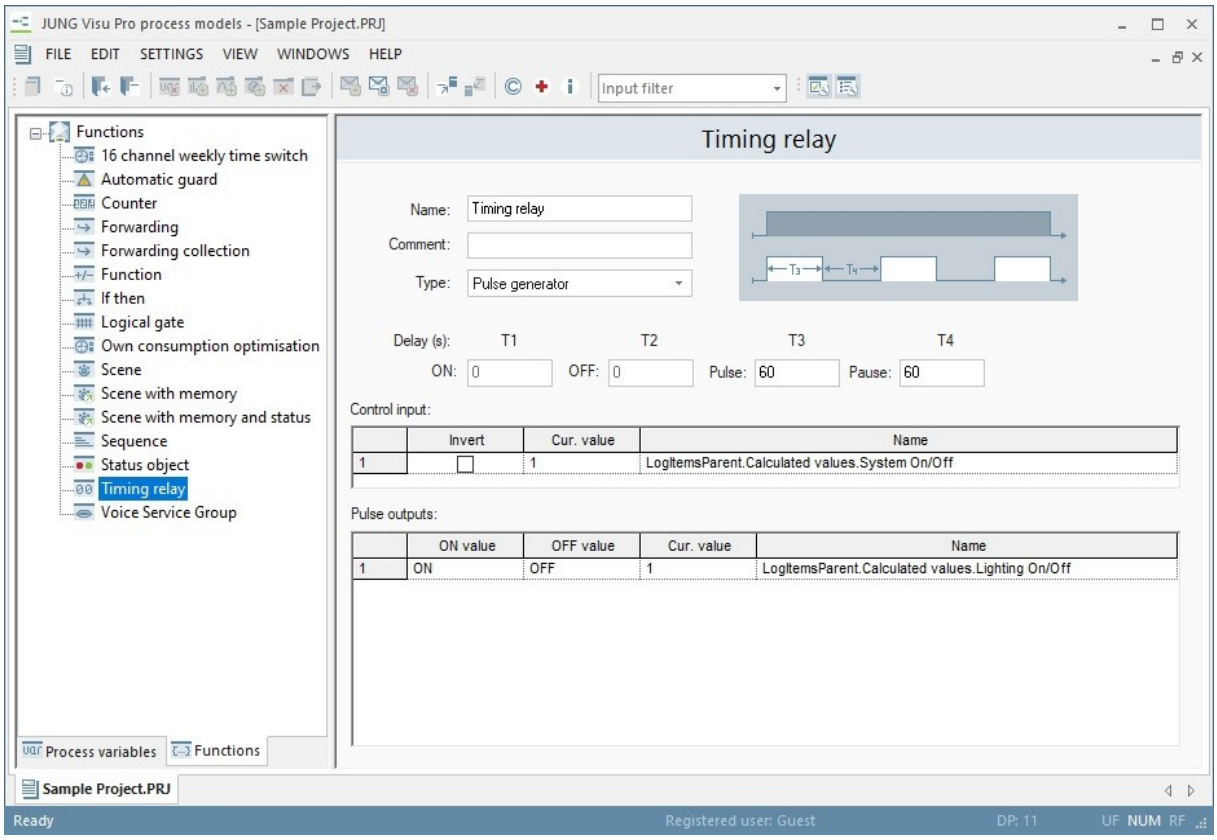
Function description:

The timing relay function can be used to cyclically send a telegram or to send an output pulse telegram with specified characteristics in return of controlling input pulse.

Configuration:

Choose the catalog tool window and select the category ***Process data***. Then drag the process variable to the grid of input or output variables.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

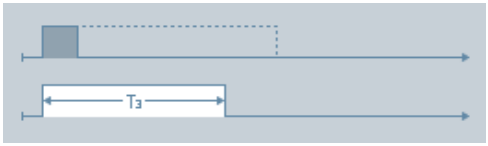
Type: Select the function type. This may be one of "pulse generator", "pulse shaper", "output delay", "input and output delay", "input wiper", "output wiper".

Delay: Specify timings in seconds: T1 ("on"), T2 ("off"), T3 ("pulse") and T4 ("pause").

The following diagrams explain the behavior of different timing relay functions depending on the type and the chosen values for the delays.

pulse generator:

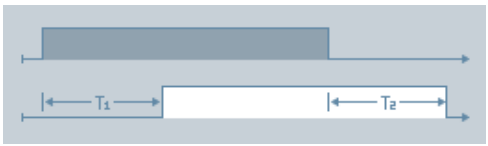
pulse shaper:



output delay:

input delay:

input and output delay:



output wiper:

input wiper:

Control input:

For function types "pulse shaper", "output delay", "input and output delay", "input wiper", "output wiper" a controlling input process variable is required. Changes on the controlling input variable will cause changes to the pulse output variables.

For the function "pulse generator", the controlling input process variable is optional. If not defined, the pulse generator will be active all the time.

Pulse outputs:

All types of process variables can be used as outputs. Alternatively to sending write telegrams for "on" and "off" pulses, it is also possible to query values with read telegrams. For binary process variables the options are provided with a combo box. For querying analog process variables enter "read" or "re" as an abbreviation.

8.16 Automatic Guard

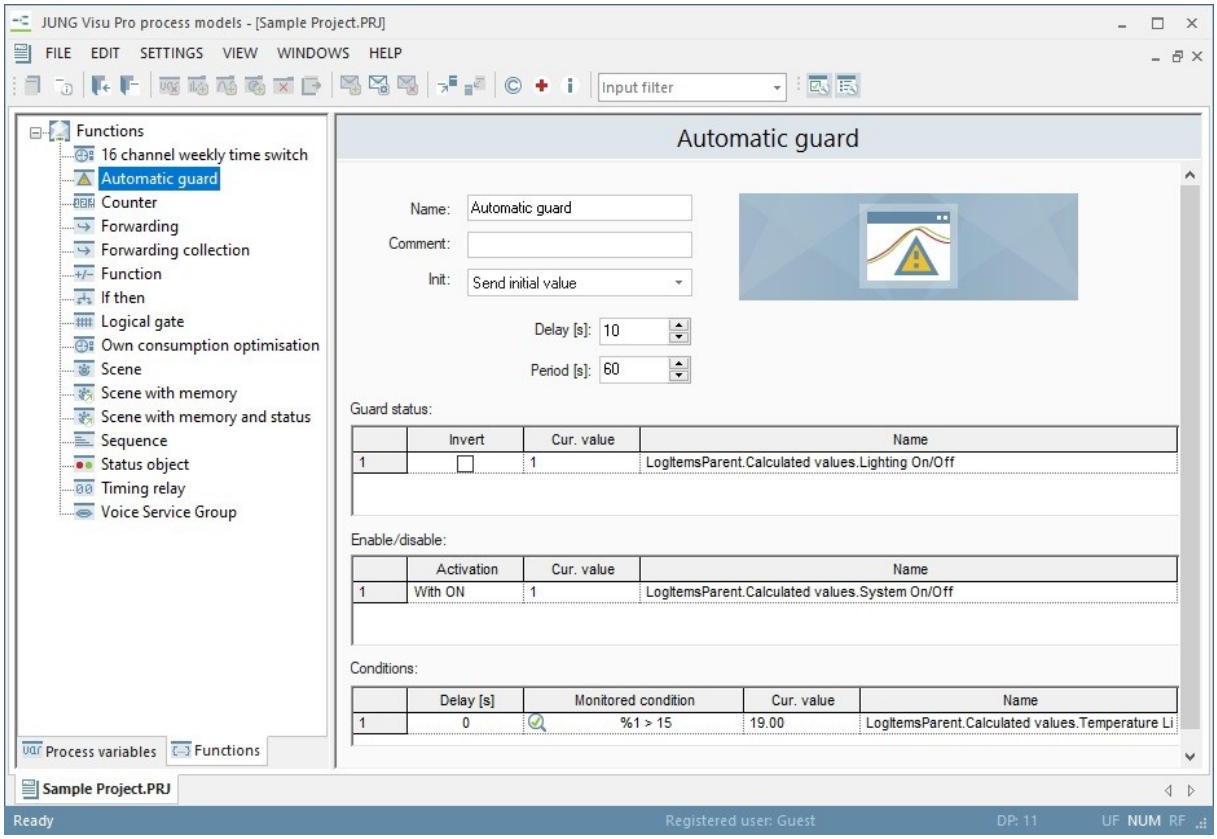
Function description:

The function *automatic guard* is used to keep a proper operating state under surveillance. The operating state is defined as a set of conditions on input variables, the conditions are defined as formulas.

Configuration:

Choose the catalog window and select the category ***Process data***. Then drag the process variable to the function window.

To delete a variable from the grid, select the row by clicking its number column. Then press the keyboard key ***Delete***.



Function settings:

Name: Enter the name of the function.

Comment: Enter a short description of the function.

Init: Allow or disallow sending the function result when the function is evaluated for the first time, i.e. after the process model has been started.

Delay: Define a delay in seconds for the evaluation of changed input values. Changes will be ignored if they are not stable for at least the delay period.

Period: Define after each time period in seconds for the evaluation of conditions.

Guard status: The binary process variable which will receive the result of the check.
1: ALARM: At least one condition is not valid.
0: OK: All conditions are valid.

Enable/disable: Optionally defines a binary process variable which will enable or disable the guard function. If "on" (value 1), the guard is active and the conditions will be evaluated, if "off" (value 0) the guard is deactivated and conditions will not be evaluated.

The process variable used for enabling/disabling the guard function may for instance be controlled by the calendar application or a switch.

Conditions:

Each row is used for one condition on one process variable. The Column ***Delay*** specifies how long the value of the input variable must be stable until the formula in the column ***Formula*** will be evaluated. In the formula use "%1" (without quotes) to refer to the process variable in the row.

Examples:

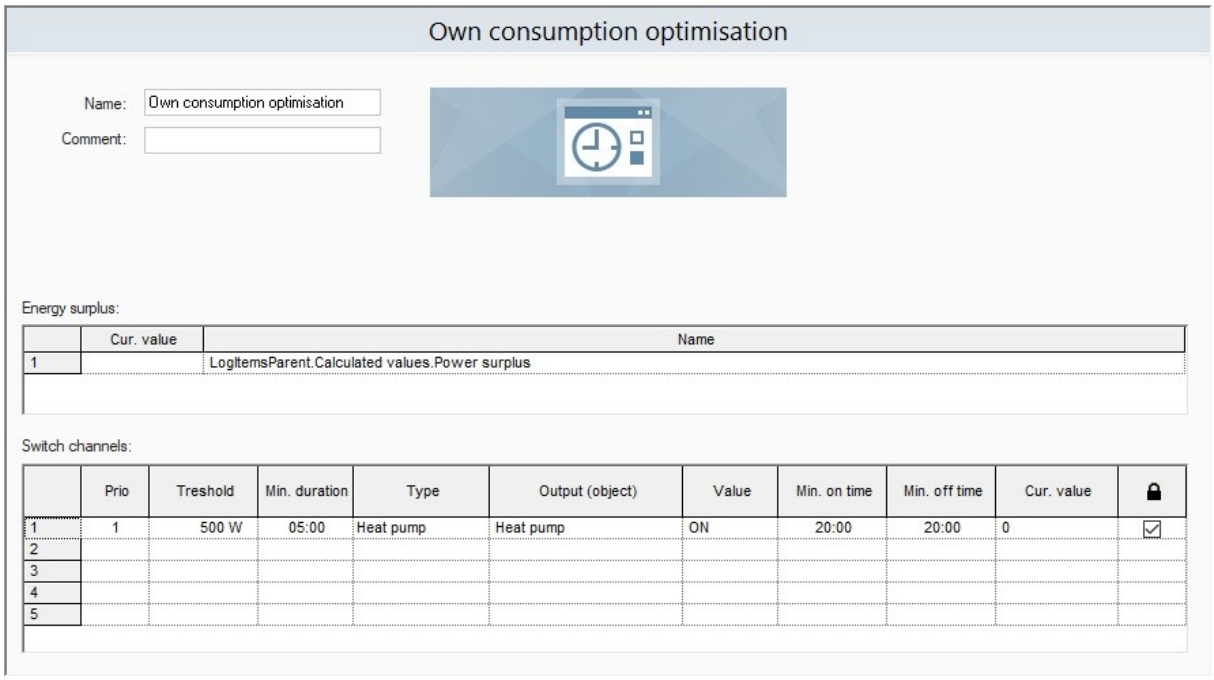
Survey temperature of cooling room:	%1 < -5
Survey bus state:	%1 = 1
Survey room temperature:	(%1 >= 18) AND (%1 <= 21)

The state of the guard will be ALARM if at least one formula returns "false" and OK if all formulas return "true".

Cur. value: This column displays the current value of the process variable.

8.17 Own Consumption Optimisation

To optimise energy consumption, the process model was extended to include the new controller function ***Own consumption optimisation***. With this new function, electrical loads can be switched depending on the energy surplus. Up to five electrical loads can be switched.



Function description:

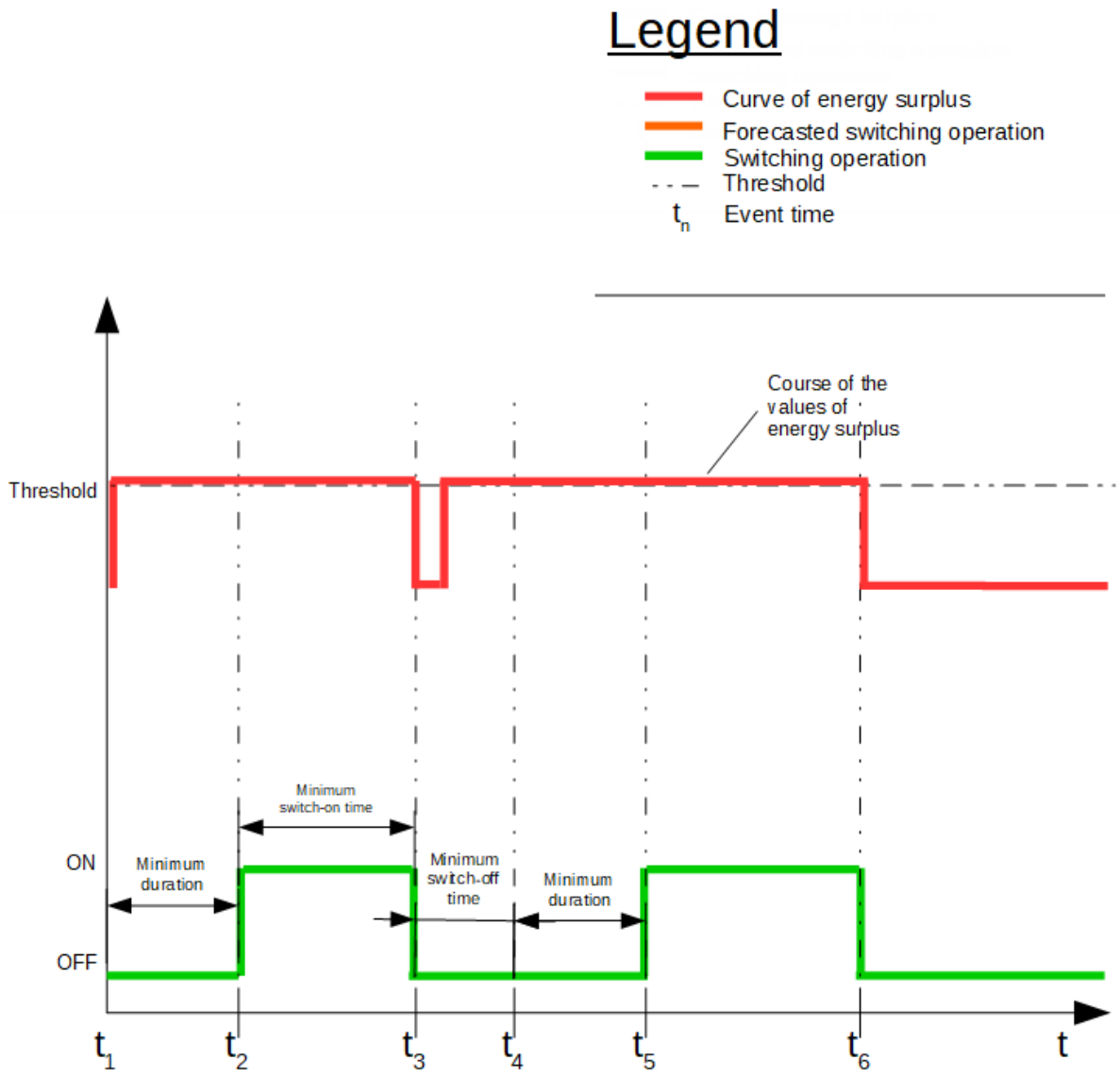
The SENEC energy storage transmits its excess energy by the analog variable linked to the optimisation function. This variable is permanently monitored by the optimisation function. If there is sufficient energy surplus, i.e. the threshold of the connected electrical load is reached or exceeded, the switch-on procedure begins.

Before switch on the electrical load, the minimum period is awaited. During this time, the value may not fall below the threshold of the energy surplus. If this is the case, this waiting period starts anew.

After expiration of the waiting period and a positive check for sufficient energy surplus, the electrical load is switched on. This is done for the period of the set minimum switch-on time. Only at the end of this period the permanent check for sufficient energy surplus continues and compares the transmitted value of energy surplus to the set threshold of the electrical load.

If the energy surplus falls below the set threshold or if the process model is closed, the optimisation function switches off the electrical load immediately.

Now the timer for the minimum switch-off time starts to run. If the value of the energy surplus rises above the specified threshold of the electrical load within this time interval, it remains switched off until waiting period has expired. A new check for sufficient energy surplus is only carried out now. If the result is positive, the electrical load is switched on again. This procedure is repeated cyclically.

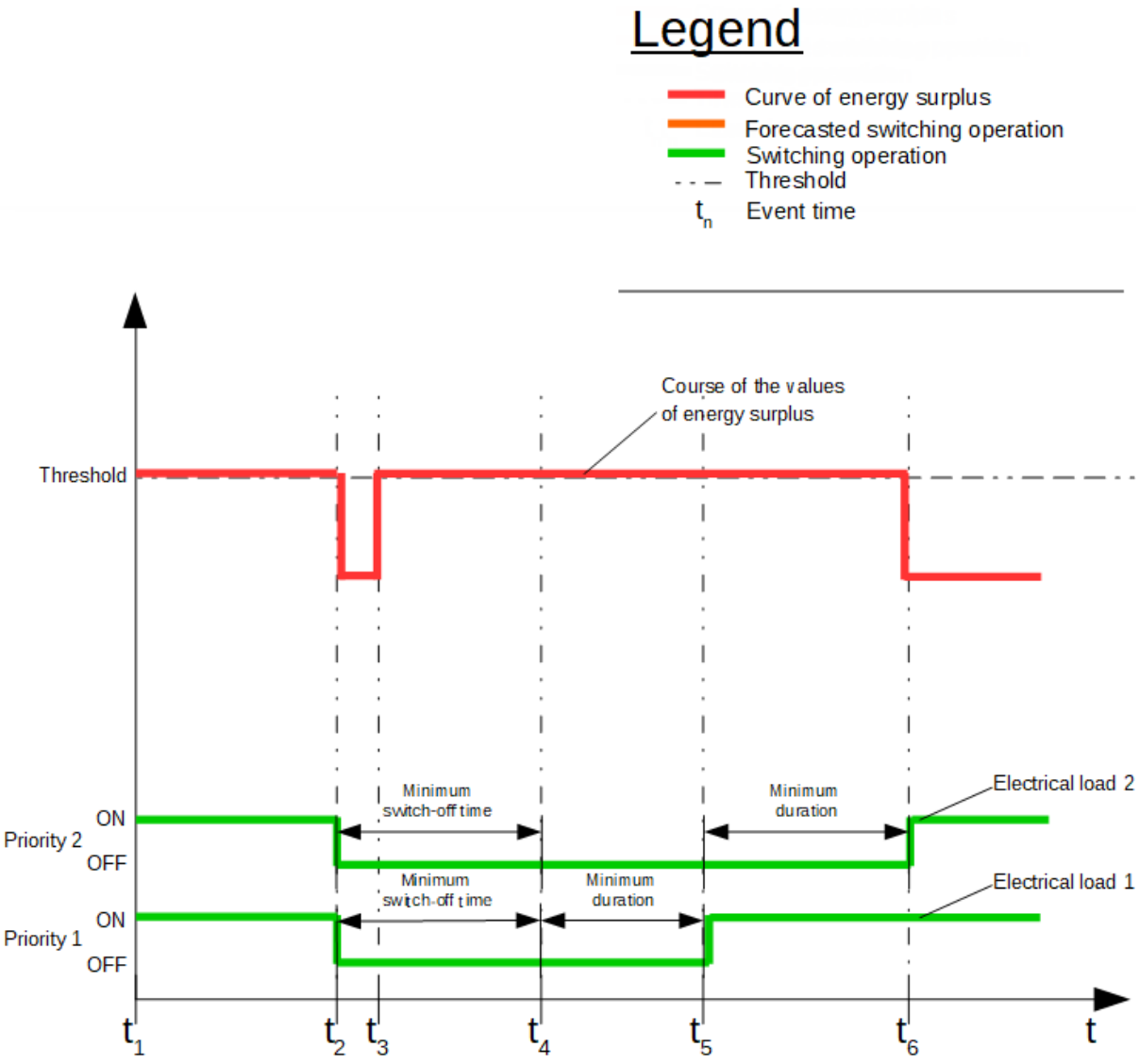

Legend
Curve of energy surplus
Forecasted switching operation
Switching operation
Threshold
t_n Event time
Course of the values of energy surplus
Threshold
Minimum switch-on time
ON
OFF
Minimum duration
Minimum switch-off time
Minimum duration
t_1
t_2
t_3
t_4
t_5
t_6
t

Function description in the case of several electrical loads:

If several electrical loads are connected, these are prioritized in order to get a sorted sequence. The electrical load with priority 1 takes priority over the electrical load with priority 2.

If the value for the energy surplus falls below the threshold of the connected electrical loads after minimum switch-on time has expired, these are switched off immediately (t2).

For these electrical loads, the timer for the minimum switch-off time begins to run now. After this interval for the higher-priority electrical load (priority 1) has expired, its timer starts for the minimum duration (t4). If there is sufficient energy surplus for the duration of this period, the electrical load is switched on again (t5) after this period has expired. All electrical loads not taken into account (priority 2 in the diagram) start anew to check the existing excess energy.



In order to optimize the switch-on and switch-off behaviour of the connected electrical loads, the electrical loads can be individually parameterized. The following parameters can be set:

Prio: If the threshold for several electrical loads is exceeded, the electrical load with the highest priority is switched on. Prio 1 is the top priority, prio 5 the lowest.

Threshold: Indicates the energy surplus in watts that must be present at least before the electrical load can be switched on.

Min. period: Determines the minimum duration in seconds that an energy surplus must be present that is higher or equal to the threshold for this electrical load.

Type: Selection of an electrical load type. Depending on the selection, parameter values are preset as well as the limits within which the parameter value may be varied.

Output (object): A binary process variable that is used to switch the connected electrical load.

Value: Binary value written by the output object when a switch-on procedure is initiated.

Min. switch-on time: The minimum duration for the switch-on procedure, even if the power surplus falls below the specified threshold.

Min. switch-off time: The minimum dwell time for the switched-off electrical load that must have expired before a new switch-on procedure can be initiated.

Cur. value: This column displays the current value of the process variable.

Lock: Configures whether or not the user of the visualization is allowed to change the parameters of the function (threshold, min. duration ...).

8.18 Voice Service Group

Function description:

The function ***Voice Service Group*** allows to control multiple datapoints with one invocation name. It can also be used like the forwarding function to switch on multiple lights with one voice command. Activating the voice service in the datapoint options before putting them into the function is not required. The group will be listed in the voice service app as a device with its name and its comment. String variables are not supported.

Every datapoint has a type allocated, which influcences the voice command over the voice service. Following types are available:

Analogue:

- Brightness
- Temperature
- Blinds

Binary:

- Turn On/Off

> ***Note:*** In a voice service group you should only set one type for analog data points (e.g. blinds **or** temperature **or** brightness). Please make sure that different types are not mixed to avoid unwanted side effects. It may not be possible to control the voice service group from the voice service interface.

Example Usage for Alexa:

Create a new voice service group with the name “Dimmer” and the comment “Dimmer in the living room”.

Add a binary datapoint to turn on/off the light and an analogue datapoint with the type “Percentage” to dim the light.

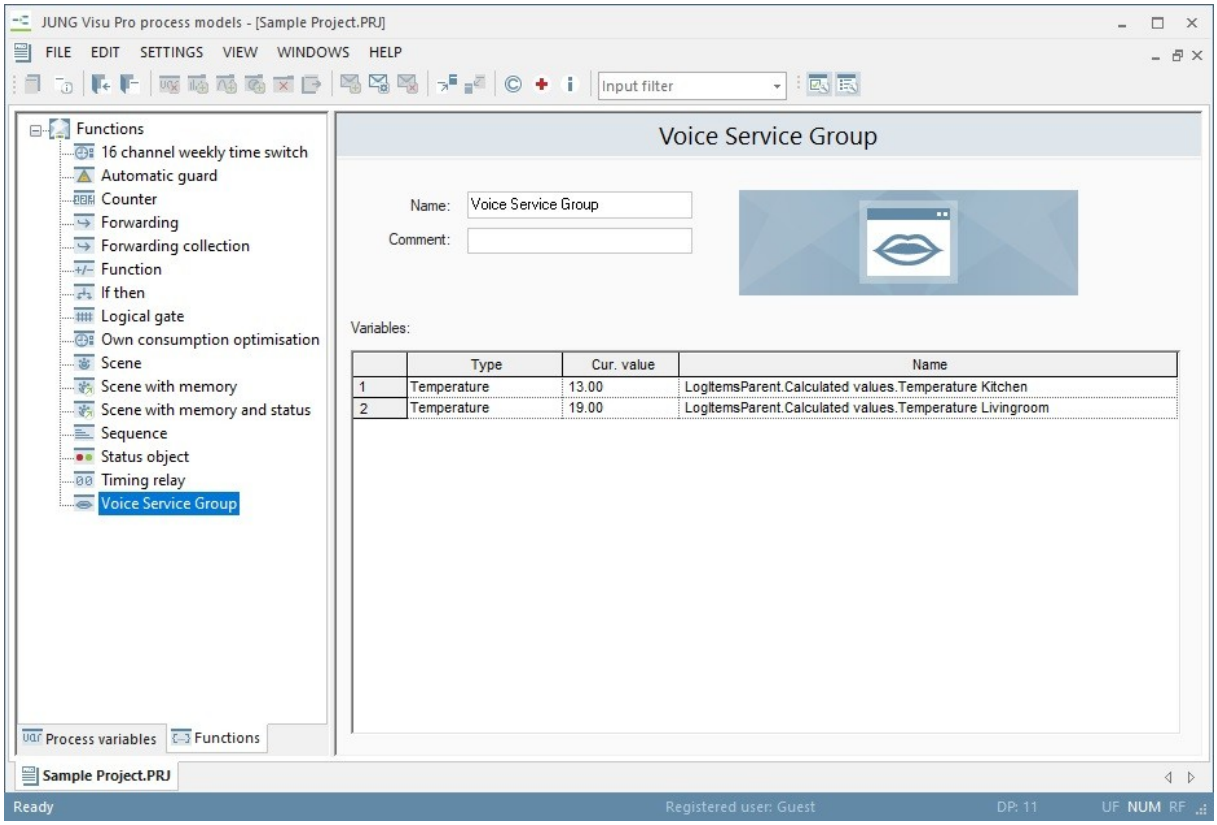
After configuring the Alexa Connection of your project you should find a device with the name “Dimmer” and the description “Dimmer in the living room” in your Alexa app. The datapoints can now be controlled with following commands:

Alexa, turn on Dimmer.
Alexa, set Dimmer to 30 percent.

8.18.1 Interpretation of the Values

Analog: For analog values, the average value from all data points located in the voice service group is calculated.

Binär: If a voice service group contains more than one binary data point, ***On*** is returned as soon as one of the data points is in the ***On*** state. If all data points are in the ***Off*** state, the voice service group also returns ***Off***.



9 Set up Voice Service

Any analog and binary process variable can be voice-operated.

9.1 Settings for the Voice Service

To use the voice service, you have to connect your JVP to your MyJUNG account, see the **visualization server manual chapter 3.1 Remote Access**.

9.2 Configure the Process Variable for Voice Control

To enable a process variable for voice control, the voice service must be activated in its settings.
A detailed description of the required entries can be found in chapter 4.7.

10 Reports

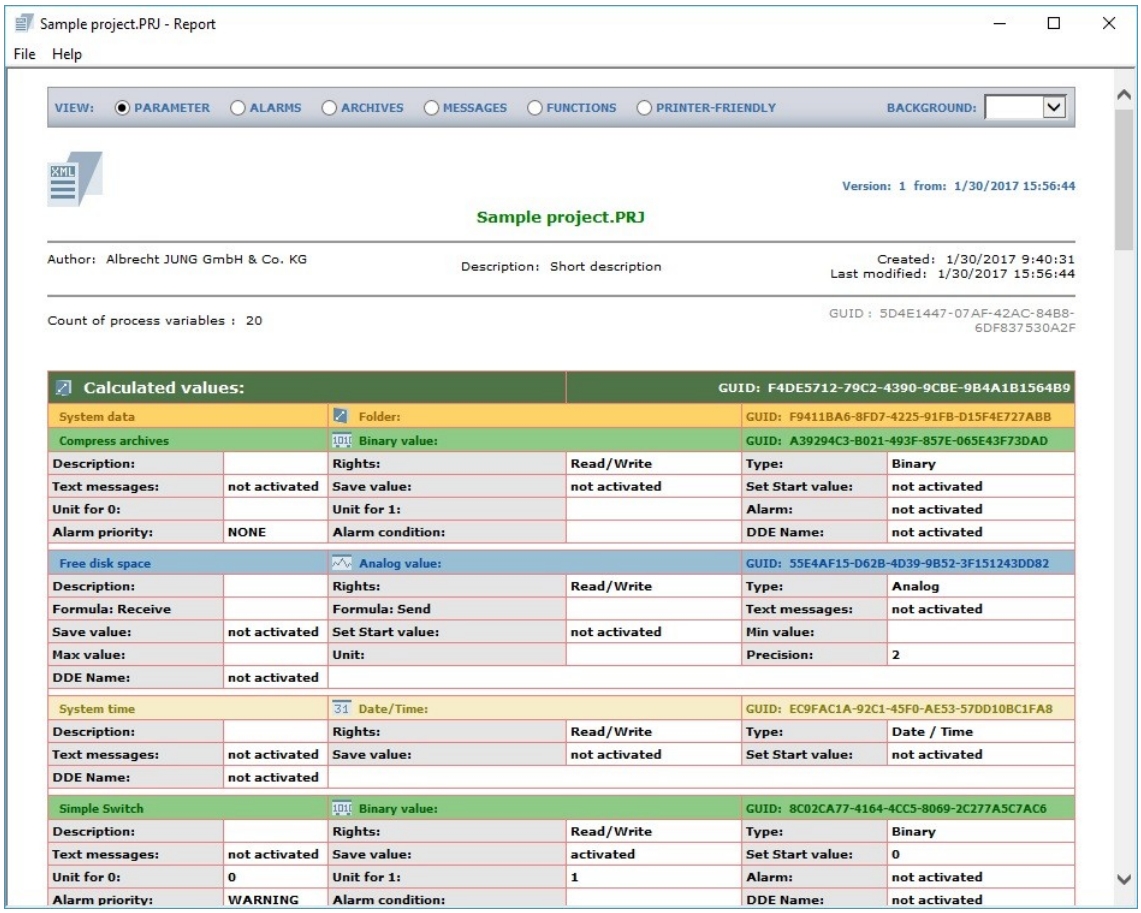
- Reports provide overviews of the project, including process variables and their properties (archives, alarms, notifications,..).
- Reports may be printed (with print preview).

10.1 Create Report

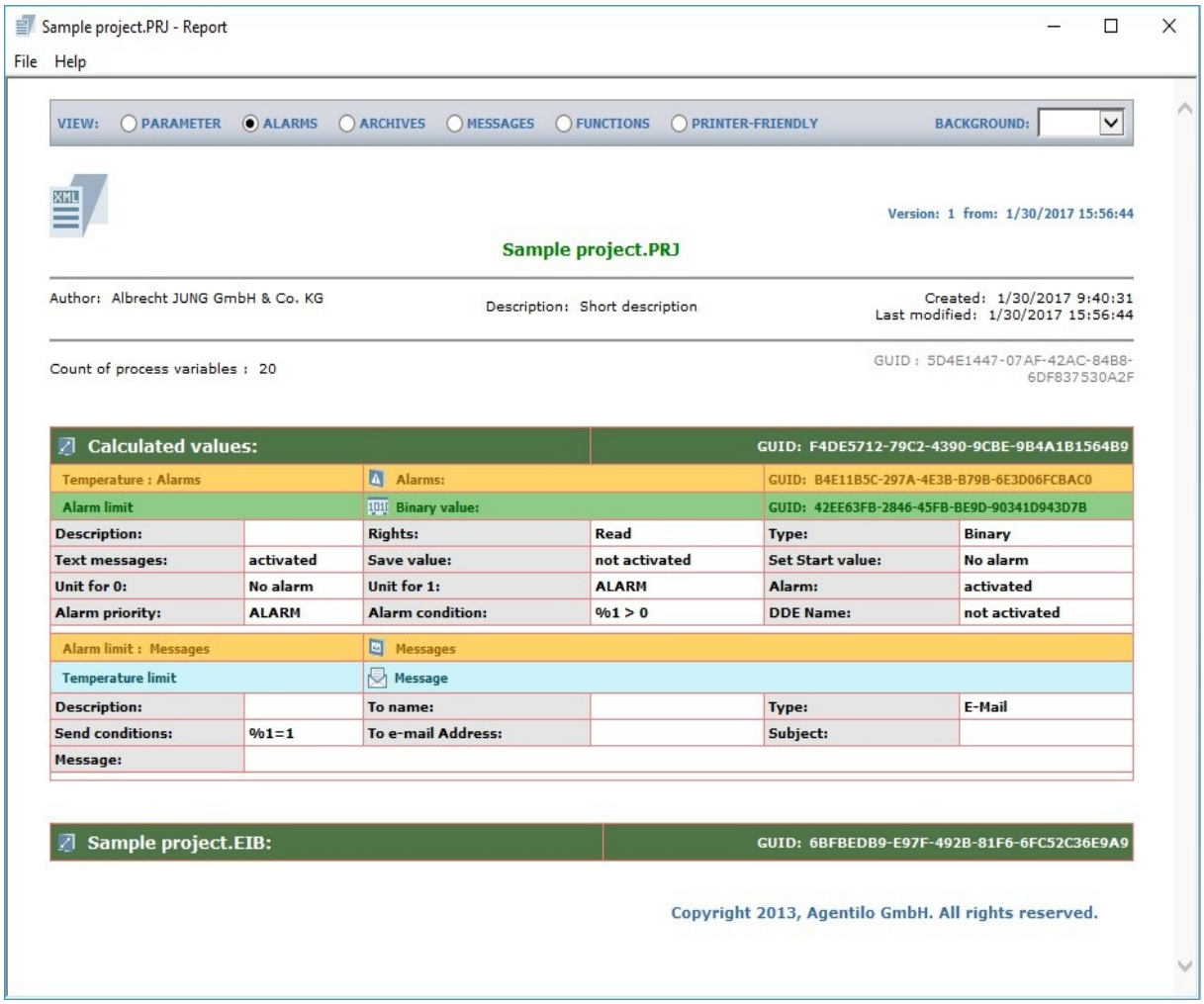
To create a report for the current project, choose the menu item ***File*** and click ***Report***. The dialog ***Create report*** opens. Select the desired options for the report generation and click the button ***Create***.

The menu at the top offers different options:

Parameter: Shows all process variables.



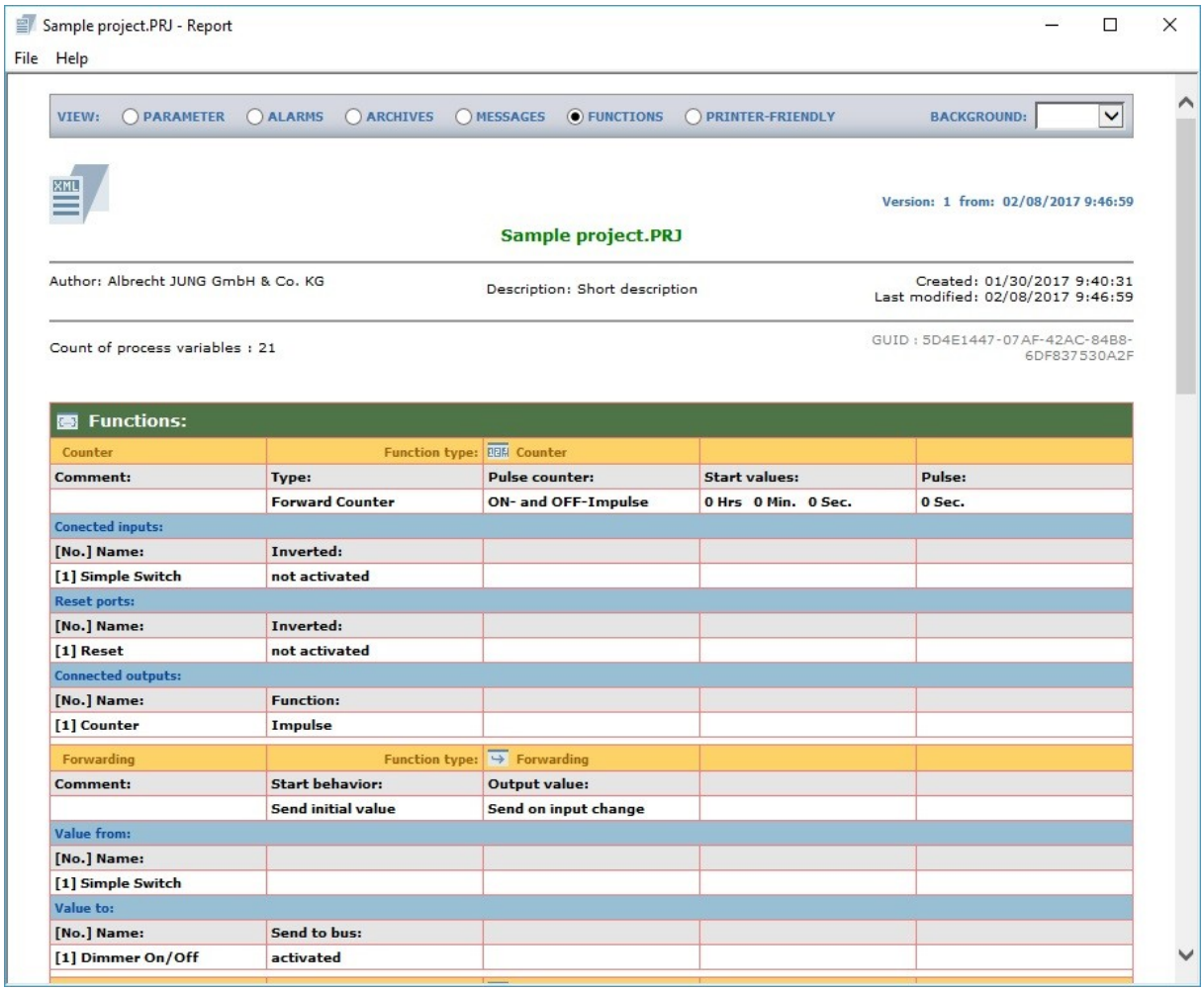
Alarms: Shows alarms and alarm conditions.



Archives: Shows archive definitions.

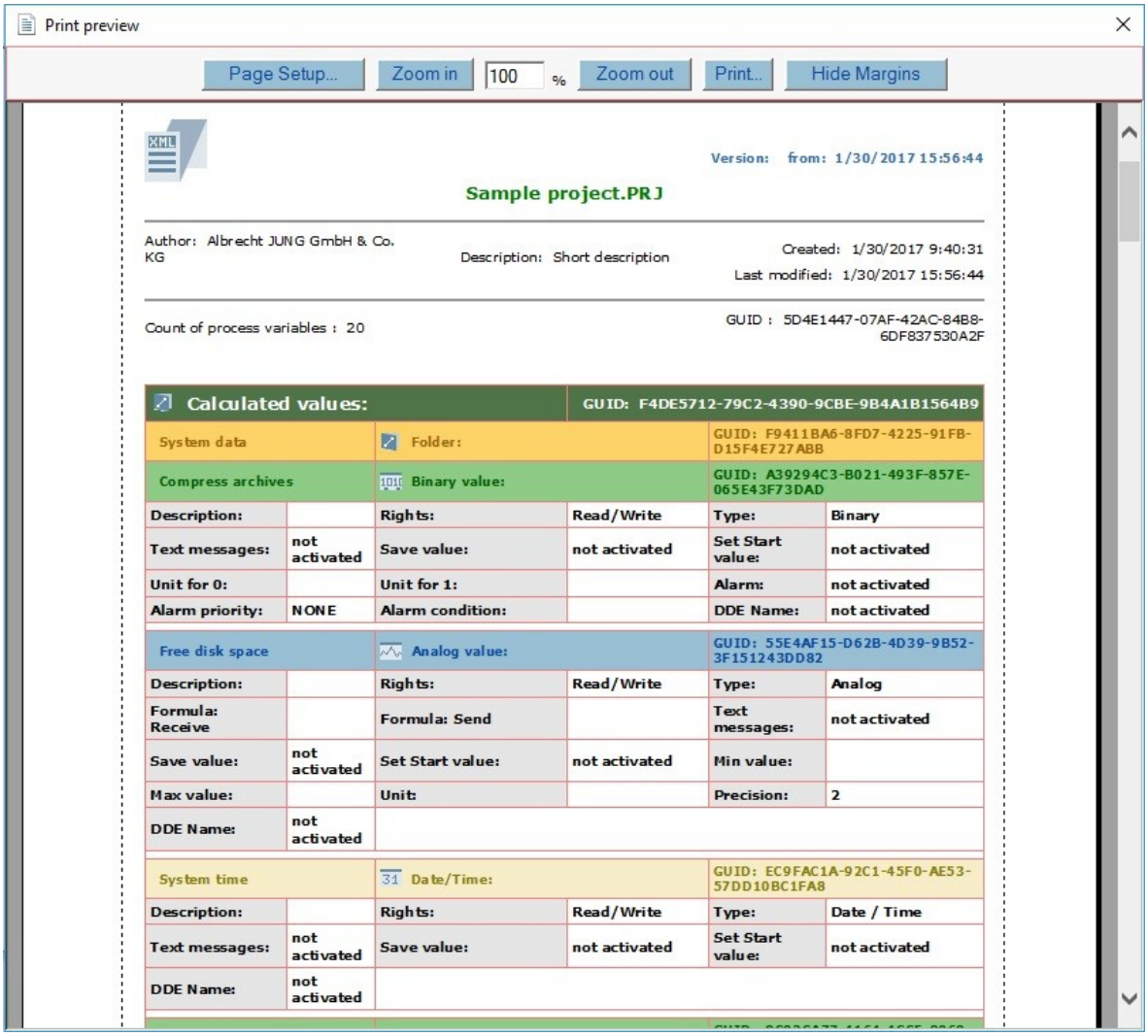
Messages: Shows configured notifications.

Functions: Shows configured controller functions.



Printer friendly: Black and white display of process variables.

10.2 Print Preview



The print preview may be scaled.

Page Setup..	Opens the dialog ***Page setup*** to set the printer properties, page properties and the alignment. Background colors and pictures can be activated or deactivated for the printing.
Zoom in	Enlarges the print preview.
Zoom out	Reduces the print preview.
Hide Margins	Button to hide the margin markers.
Show Margins	Button to display the margin markers.

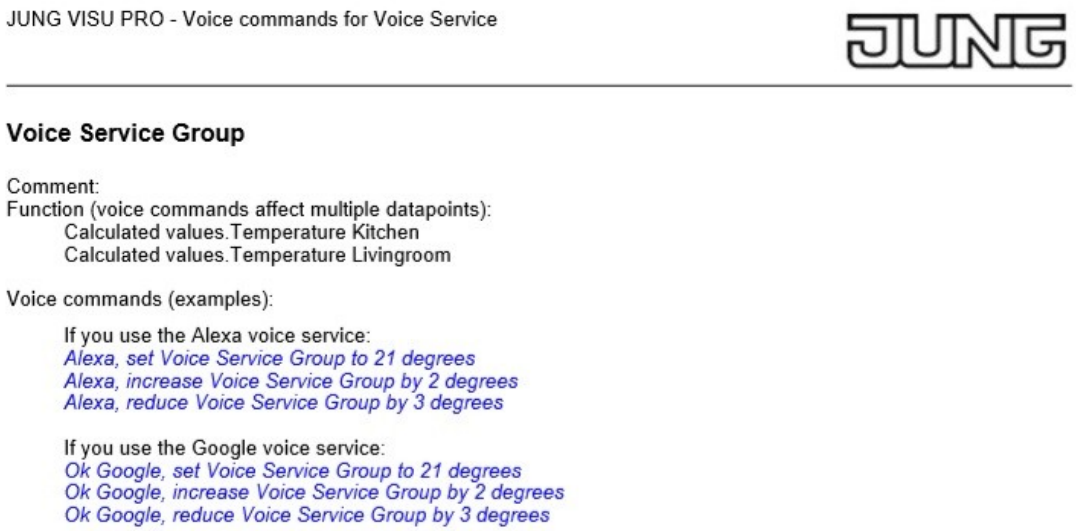
With these , markers the user has visual control over the margin settings.

10.3 Voice Service Report

It is possible to create a new PDF report in the menu ***Settings*** under ***Voice service report***. This report shows all datapoints, which are activated for the voice control, as well as every voice service group in the project.

For each datapoint / voice service group the report contains following information:

- The name, which is used in the device list of the voice assistant to identify the datapoint / voice service group.
- The comment, which is used as the description in the device list.
- The datapoints, which are controlled with that name.
- Examples for voice commands, which can be used to control the datapoints / voice service group.



JUNG VISU PRO - Voice commands for Voice Service

Voice Service Group

Comment:
Function (voice commands affect multiple datapoints):
Calculated values.Temperature Kitchen
Calculated values.Temperature Livingroom

Voice commands (examples):

If you use the Alexa voice service:
Alexa, set Voice Service Group to 21 degrees
Alexa, increase Voice Service Group by 2 degrees
Alexa, reduce Voice Service Group by 3 degrees

If you use the Google voice service:
Ok Google, set Voice Service Group to 21 degrees
Ok Google, increase Voice Service Group by 2 degrees
Ok Google, reduce Voice Service Group by 3 degrees

JUNG Visu Pro
Visualization Editor

Content:

1 TECHNICAL REQUIREMENTS.....5

1.1 Hardware.....5
1.2 Operation Systems.....5
1.3 Internet Explorer, DirectX and .NET Framework.....5

2 FIRST STEPS.....6

2.1 Step 1: Create Visualization Project.....6
2.2 Step 2: Create Worksheet.....8
2.3 Step 3: Import from Process Model.....9
2.4 Step 4: Insert a Display Element.....12
2.5 Step 5: Connect a Display Element to a Process Variable.....13

3 USER INTERFACE.....15

3.1 Menu.....16
3.2 Toolbar.....20
3.3 Tool Windows.....21
3.3.1 Tasks.....21
3.3.2 Properties.....22
3.3.3 Elements.....24
3.3.4 Catalog.....27
3.4 Window Management.....29
3.5 The Library.....32

4 FUNCTIONS.....34

4.1 Project Organizer.....34
4.2 Project Properties.....40
4.3 Worksheet Organizer.....44
4.4 Worksheet Properties.....45
4.4.1 Define Resolution.....45
4.5 Copy the Worksheet.....46
4.6 Edit the Worksheet.....47
4.6.1 Define Background.....47
4.6.2 Insert Text Item.....49
4.6.3 Insert Image Item.....50
4.6.4 Animated Images.....51
4.6.4.1 Image Library.....51
4.6.4.2 Example: Analog Image Output.....51
4.6.5 Using Grid and Guides.....53
4.6.6 Moving Elements with Keyboard Keys.....55
4.7 Checking Functions.....56
4.7.1 Check Project.....57

5 ELEMENTS.....59

5.1 Panels....59
5.1.1 Style Options....61
5.1.2 Border Panel....62
5.1.3 Main Panel....63
5.1.4 Stack Panel....64
5.1.4.1 SCIP Stack Panel....64
5.1.5 Tab Panel....65
5.2 Controls....66
5.2.1 General Settings....66
5.2.1.1 Tab *General*....67
5.2.1.1.1 Element Name....67
5.2.1.1.2 Label....68
5.2.1.1.3 Layout....68
5.2.1.1.4 Mode of Usage....68
5.2.1.2 Tab *Detail Settings*....69
5.2.1.2.1 Enable....69
5.2.1.2.2 Presetting....69
5.2.1.2.3 Effects....70
5.2.1.2.4 Selection Group....70
5.2.1.3 Tab *Display*....71
5.2.1.4 Tab *Value Display*....*72*
5.2.1.5 Tab *Connections*....77
5.2.2 Control Properties....78
5.2.2.1 Switch....78
5.2.2.2 Blind....79
5.2.2.3 Slat....81
5.2.2.4 Dimmer....81
5.2.2.5 Multipurpose Control....83
5.2.2.6 Control Send Fixed Value & Control Step Value....85
5.2.2.7 Slider....86
5.2.2.8 Button....94
5.2.2.8.1 Show Worksheet....94
5.2.2.8.2 Toggle Tab Panel....95
5.2.2.8.3 Set Chart Property....95
5.2.2.8.4 Toggle Main Panel....96
5.2.2.8.5 Standard Link....97
5.2.2.8.6 Open Link in Window/Tab....97
5.2.2.8.7 HTTP GET Request....97
5.2.2.8.8 Back....97
5.2.2.8.9 Toggle Embedded Site....98
5.2.2.8.10 Logout HTML Client....98
5.2.2.8.11 Show Worksheet by Numerical Code....98
5.2.2.8.12 Alexa Nameconfiguration....99
5.2.2.9 Basic Setpoint....100
5.2.2.10 Basic Setpoint Shifting....100
5.2.2.11 Operation Modes....102
5.2.2.12 Operation Mode Switch Over....103
5.2.2.13 Multipurpose Display....104
5.2.2.14 Image Display....105
5.2.2.15 Binary Image Input....107
5.2.2.16 Binary Image Output....108
5.2.2.17 Analog Image Output....109
5.2.2.18 Text Input....110
5.2.2.19 Text Input/Output....111
5.2.2.20 Numeric Value Input/Output....111
5.2.2.21 Date/Time Input....111
5.2.2.22 Timestamp....111

5.2.2.23 Stand-Alone Slider....112
5.2.2.24 Slider with Display....114
5.2.2.25 Double Stand-Alone Slider with Blind / Slat Display....115
5.2.2.26 Color Input....116
5.2.2.27 Colour Control....117
5.2.2.28 Tunable White Control....118
5.2.2.29 Knob....119
5.2.2.30 Dimmer Knob....120
5.2.2.31 Embedded Site....121
5.2.3 16 Channel Weekly Time Switch....122
5.2.3.1 Control Settings....122
5.2.3.2 Operation via HTML User Interface....122
5.2.3.2.1 Column Contents....123
5.2.3.2.2 Create and *C*onfigure *S*witching *C*ommands....124
5.2.3.2.3 Create and *C*onfigure *Astro C*ommands....124
5.2.3.2.4 Creating a Switching Command....125
5.2.4 16 Channel Store Scene with ACK....129
5.2.4.1 Control Settings....129
5.2.4.2 Operation via HTML User Interface....129
5.2.4.2.1 Column Contents....130
5.2.4.2.2 Using of the Overlay....131
5.2.5 Charts....132
5.2.5.1 Configuring the Display Element within the Visualization Editor....132
5.2.5.2 Handling the HTML User Interface....134
5.3 SENEC-Specific Controls....135
5.3.1 Connect SENEC Controls to Process Variables....135
5.3.2 General Settings....137
5.3.2.1 Tab *General*....137
5.3.2.1.1 Element Name....137
5.3.2.1.2 Label....138
5.3.2.1.3 Values....138
5.3.2.2 Tab *Detail Settings*....139
5.3.2.3 Tab *Display*....139
5.3.2.4 Tab *Overlay-Layout*....139
5.3.2.4.1 Properties....139
5.3.2.4.2 Handling the HTML User Interface....141
5.3.2.5 Tab *Connections*....143
5.3.3 Element Properties....144
5.3.3.1 Power Consumption....144
5.3.3.2 Battery Charging....144
5.3.3.3 Mains Current Feed-in....144
5.3.3.4 Battery Level....145
5.3.3.5 Power Generation....145
5.3.3.6 Battery Draw....145
5.3.3.7 Mains Current Draw....145
5.3.3.8 Total Savings....146
5.3.3.9 SENEC Control....146
5.3.3.9.1 Handling the HTML User Interface....147
5.3.3.10 Energy Traffic Light....148
5.4 Picture Selection....149
5.4.1 Symbolic Path: @sys....149
5.4.2 Symbolic Path: @proj....149
5.4.3 Animated Images....150

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per Pixel.
Interfaces	Serial or USB interface to connect KNX with the FALCON driver.

1.2 Operation Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Internet Explorer, DirectX and .NET Framework

Microsoft Internet Explorer, DirectX version 9.0c and .NET Framework 3.5 Service Pack 1 are required. The latest versions can be installed or updated from the setup CD.

2 First Steps

After the installation of the visualization system a program group is created in the Start Menu. The default settings can be adopted or manually changed during installation.

To start the control panel of the visualization, click the program icon. In order to clearly distinguish between the different applications, different colors were used. The icon of the **visualization editor** contains a yellow colored bar .

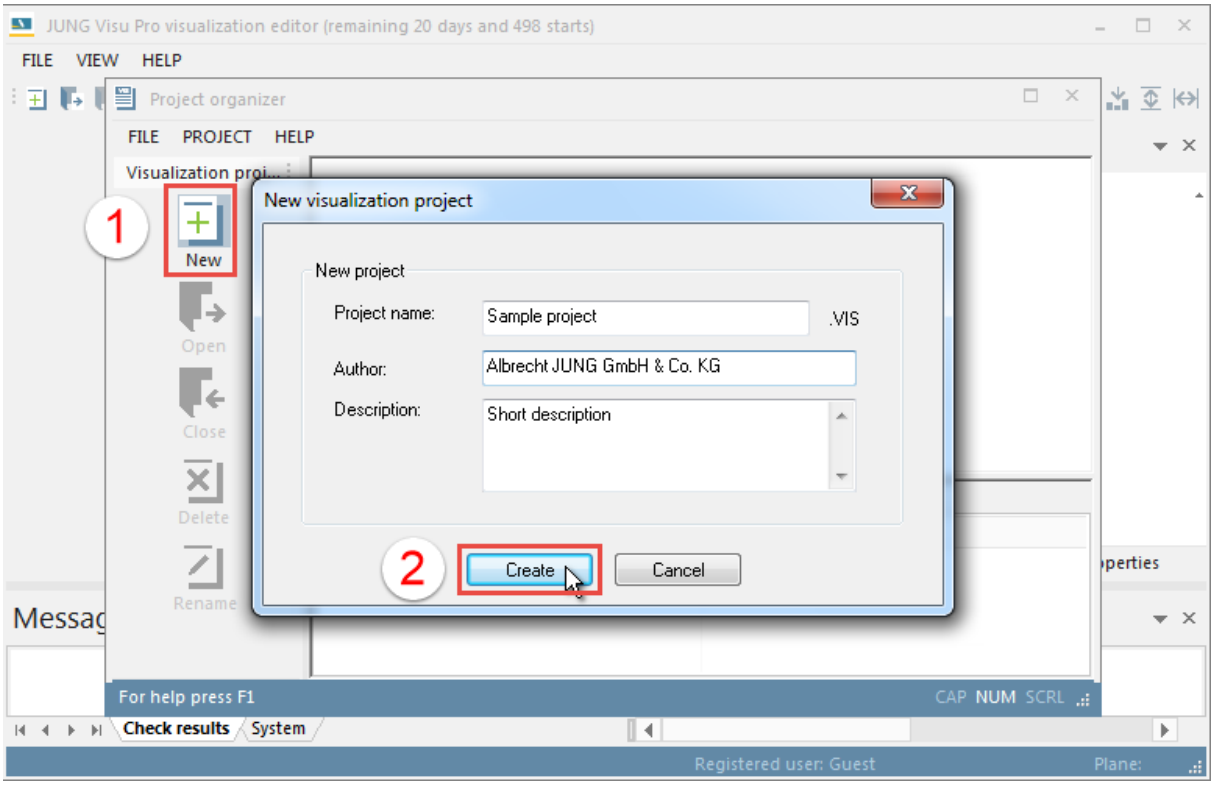
In the tab *function modules* you start the application with a double click on the icon of the respective module.

The visualization editor is a companion of the visualization server, which is used to display the process images.

2.1 Step 1: Create Visualization Project

To create a new project, start the dialog ***Project Organizer***. In the Project Organizer you can open, close, rename or delete projects. To open the Project Organizer, choose the menu entry ***Projects*** or click the symbol ***Organize*** in the tasks tool window.

Click the symbol ***New***. The dialog ***New visualization project*** opens.



Enter the data of the new project:

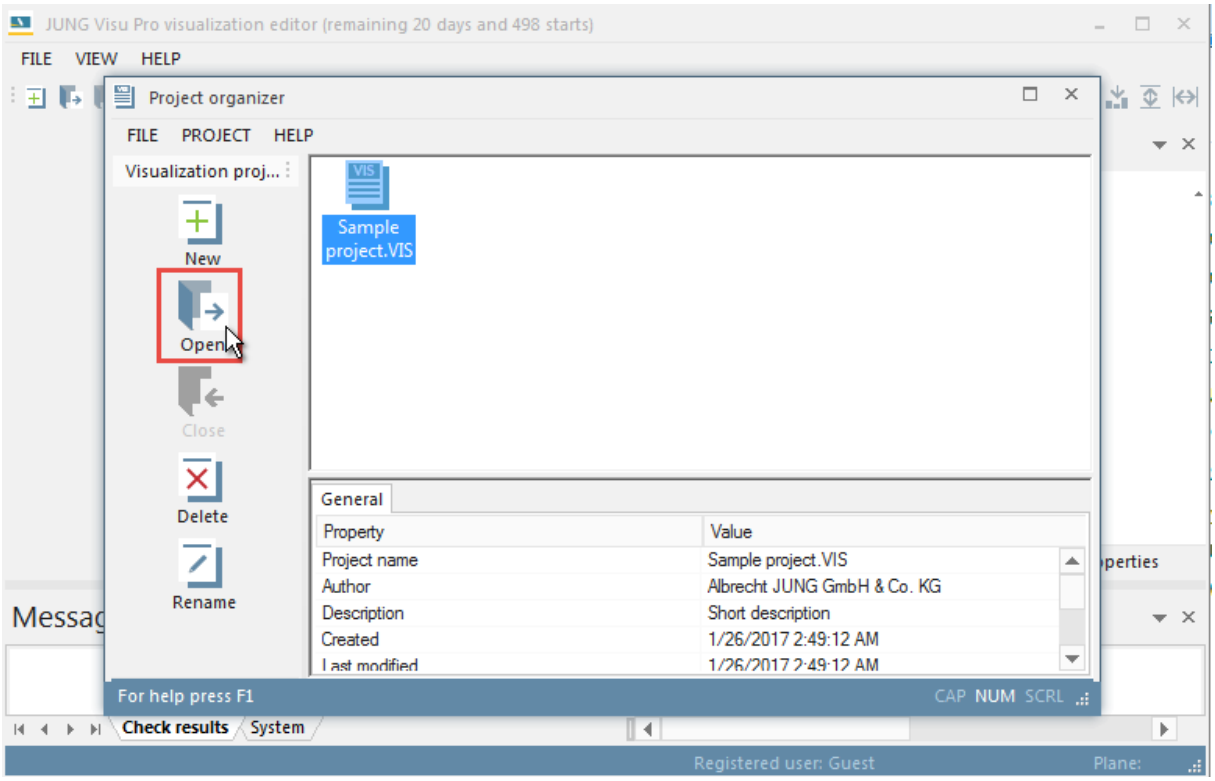
Project name: Name of the visualization project.

Author and description: It is useful but not mandatory to enter the name of the author or the project description. These entries can be changed later.

After entering at least the name of the project, click the button ***Create*** to confirm the creation of the project. The project is now contained in the Project Organizer and can be started from here.

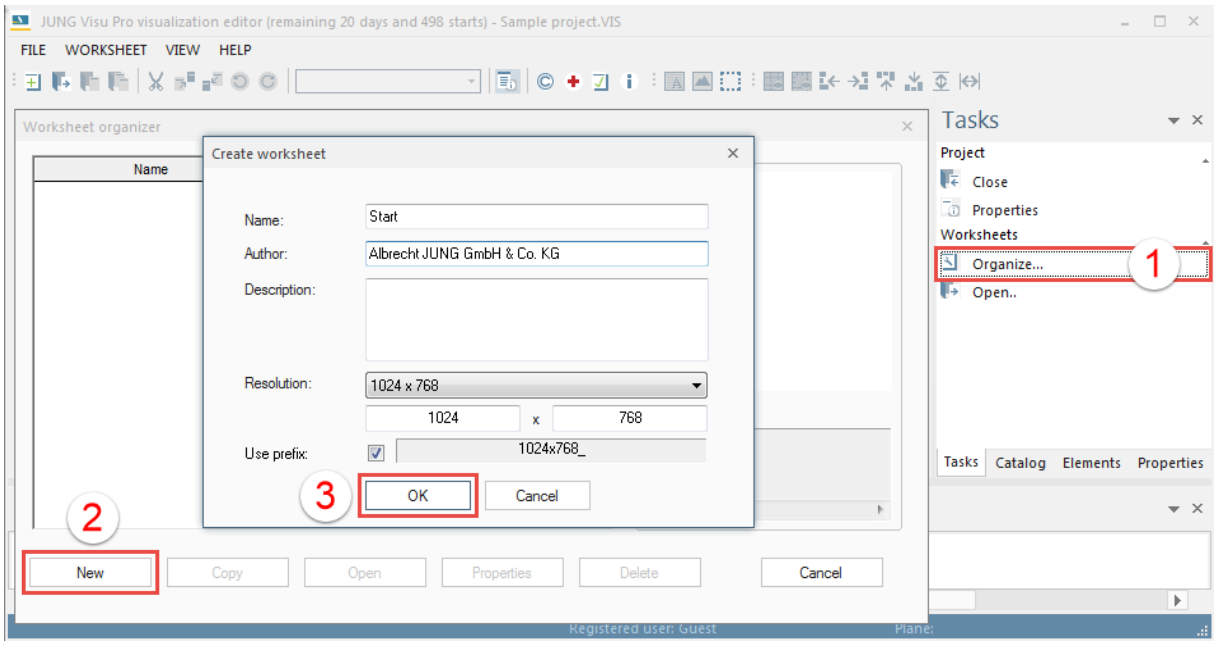
To open the project in the Project Organizer:

- Double-click the project icon or
- select the project and click the icon ***Open*** in the sidebar or
- select the project, choose the menu entry ***Project*** and click ***Open***.



2.2 Step 2: Create Worksheet

Choose the menu entry ***Worksheet*** and click on ***Worksheets***. Alternatively, click the icon ***Organize*** in the tasks tool window. The dialog ***Worksheets*** opens where you can create new worksheets and organize existing ones. Click here on the button ***New***. The dialog ***Create Worksheet*** opens.



Enter the data of the new worksheet:

Name: Enter the name of the worksheet (required).

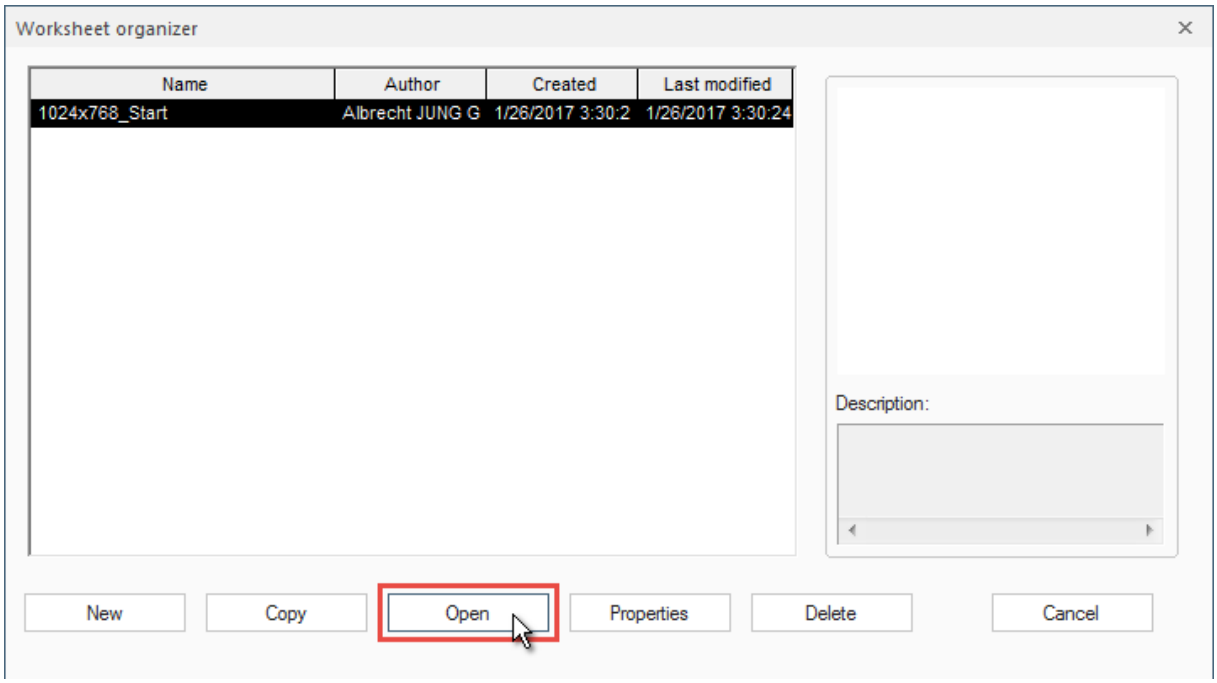
Author and description: It is not mandatory but useful to enter the name of the author and a short description. These data can be added later.

Resolution: Screen resolution for the worksheet.

Use prefix: Sets the screen resolution as a prefix in front of the worksheet name. Only worksheets with prefix are detected by the server.

Click the button ***OK***. The worksheet is now created.

To open the worksheet in the editor, select it and click the button ***Open***.

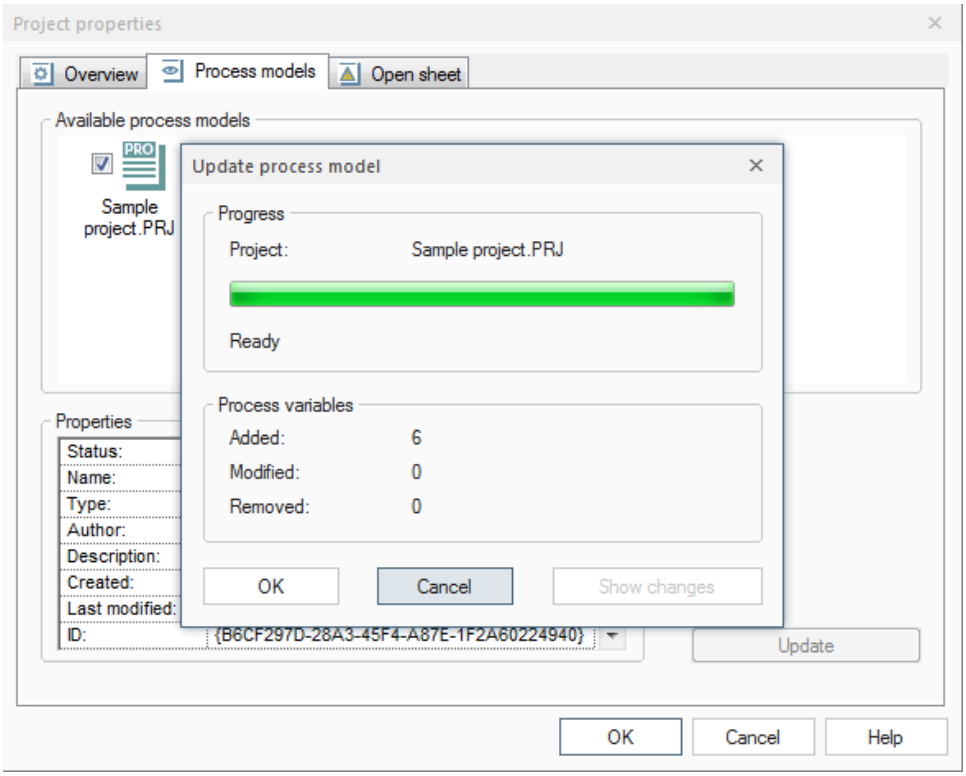


Click on ***Cancel*** to close the dialog ***Worksheet organizer***.

2.3 Step 3: Import from Process Model

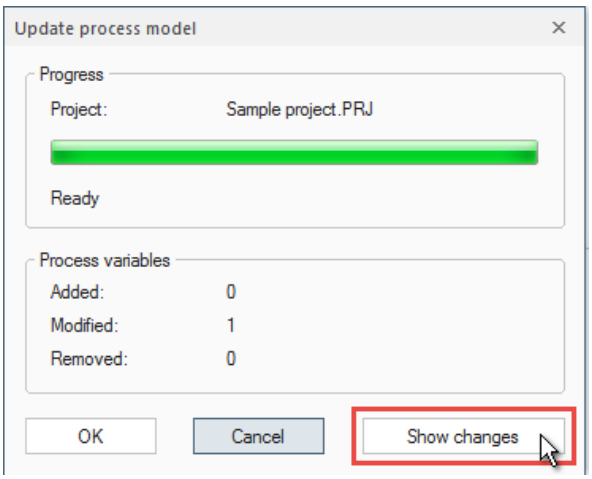
Choose the menu option ***File*** and click on ***Project properties***. The dialog ***Project Properties*** opens.

Choose the tab ***Process models*** which contains a list of all the available process models. To connect the visualization project with a process model, select the appropriate check box in front of the process model icon. The dialog ***Update Process Model*** opens and the update starts automatically.



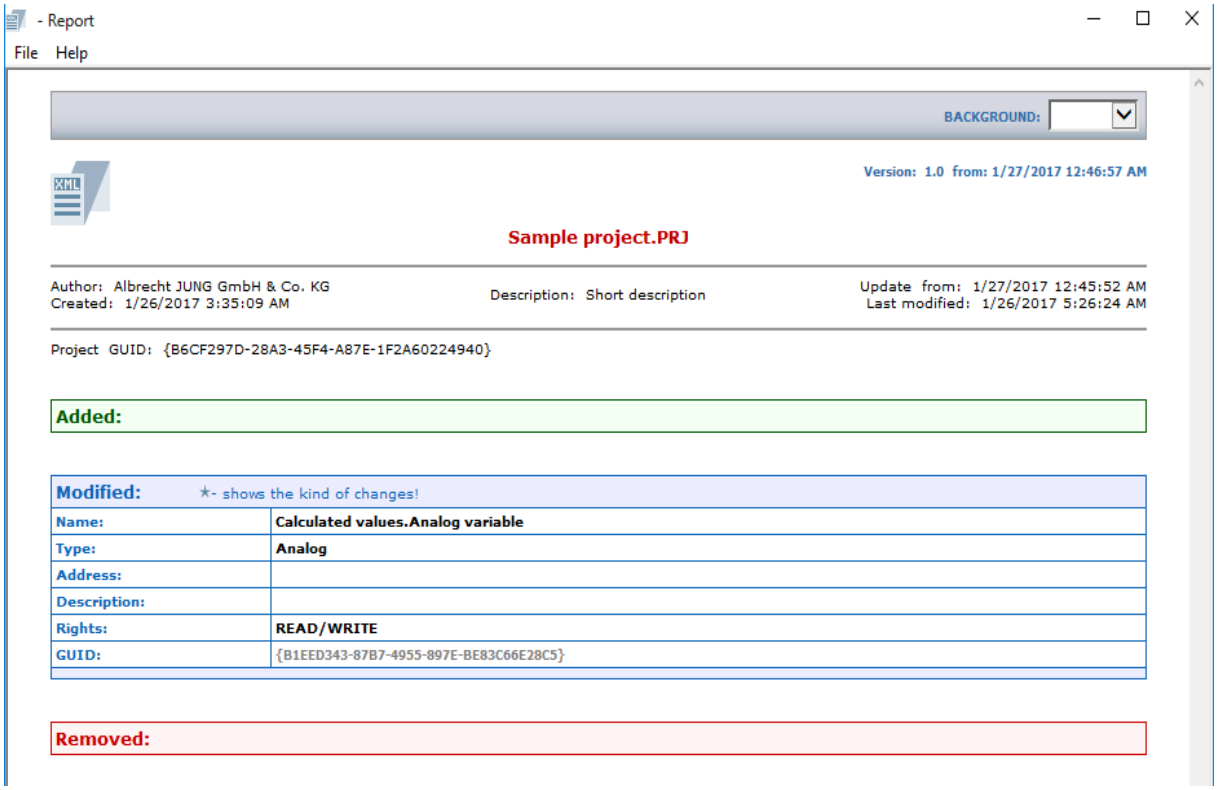
If the logical process model is imported for the first time, the button ***Changes*** is dimmed. After the update of the process model has been finished, the number of the added process variables is indicated in the dialog.

If the process model has been modified, the button ***Changes*** is enabled.



A click on it opens a report comprising the modifications carried out on the process variables (see below).

The report can be printed out and used later to compare the data with the project report. Thus, modifications carried out in the visualization project can be easily seen.



New process variables are displayed in green color.

Modified process variables are displayed in blue color. The modified properties are marked with an asterisk.

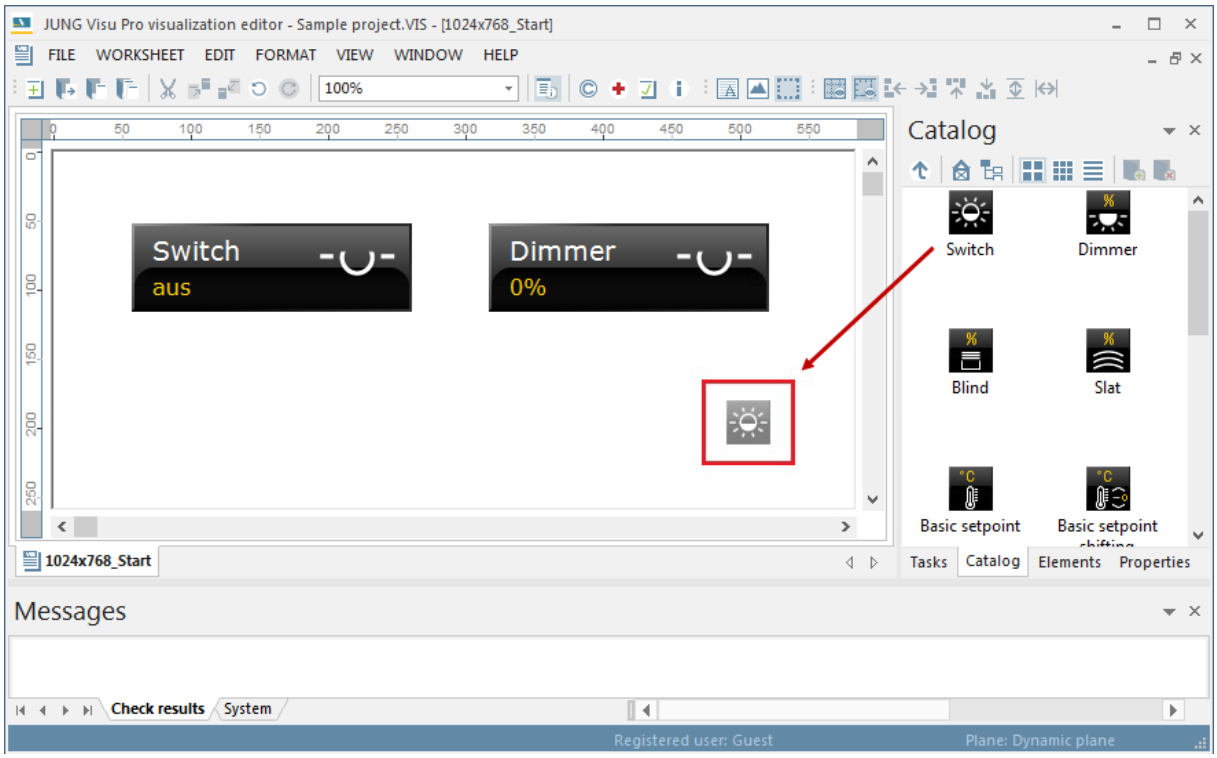
Deleted process variables are displayed in red color.

2.4 Step 4: Insert a Display Element

Choose the tool window ***Catalog*** and click on the object group ***Controls***. Here you can find the different types of display elements which can display values of process variables.

The display elements are grouped in different directories. For example a double click on the directory ***Binary*** will show switches and other display items, which can be used to set a binary value in the technical process.

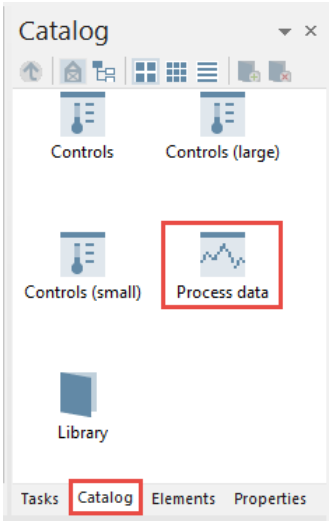
To insert a display element into the worksheet, select the element by pressing the left mouse button, hold down the mouse button and drag the element into the worksheet. Release the mouse button to drop the element at the current mouse position. The properties dialog opens where you can define the properties of the element. Further details of the display elements can be found in chapter 5.2.



Save the worksheet by clicking the button in the toolbar.

2.5 Step 5: Connect a Display Element to a Process Variable

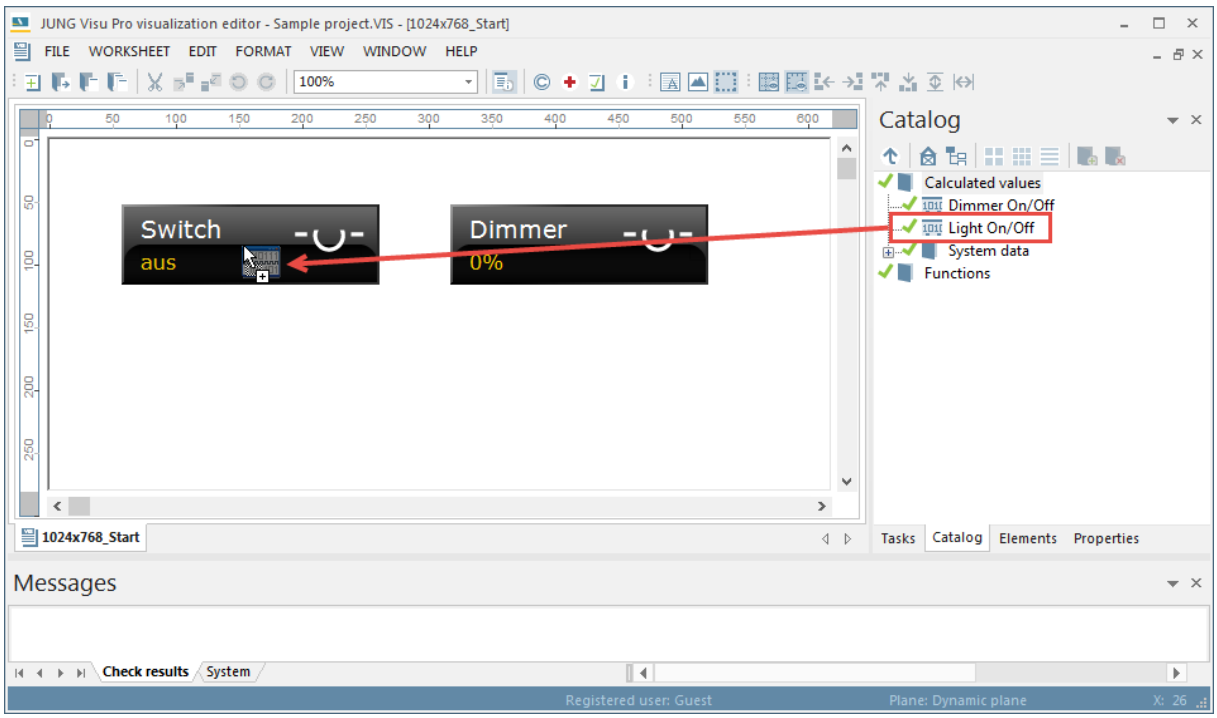
To connect a process variable to a display element, a process model must be created and connected to the visualization project.



Choose in the tool window ***Catalog*** the object group ***Process Data*.** Here you can browse the tree of process variables.

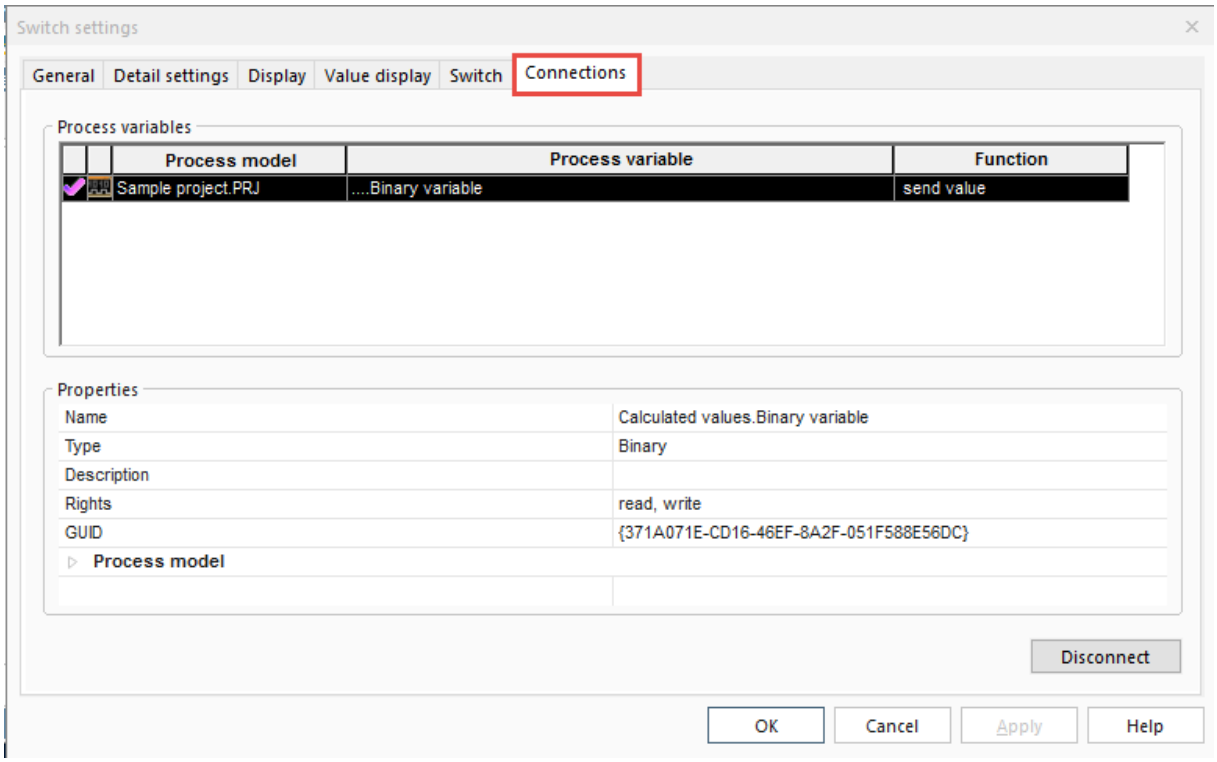
Connecting a display element to a process variable:

In the process variable tree select the process variable you wish to connect to the display element.

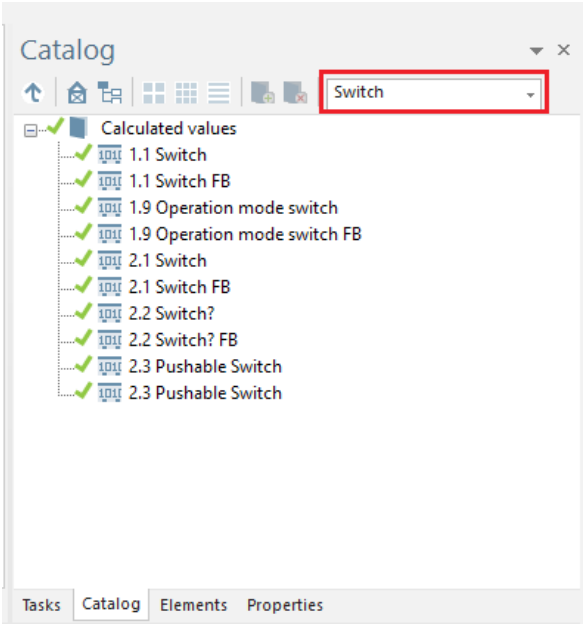


Then drag the process variable to the display element. If the process variable has been dropped successfully, a connection between the display element and the process variable has been established. Save the worksheet.

The process variables connected to the display element are listed on the tab ***Connections*** in the properties dialog of the display element. Double-click the display element to open the properties dialog (see below).



Note: To filter the process variables or functions, click into the search box ***Enter Filter*** and enter a search term. Then press the ***Enter*** key.



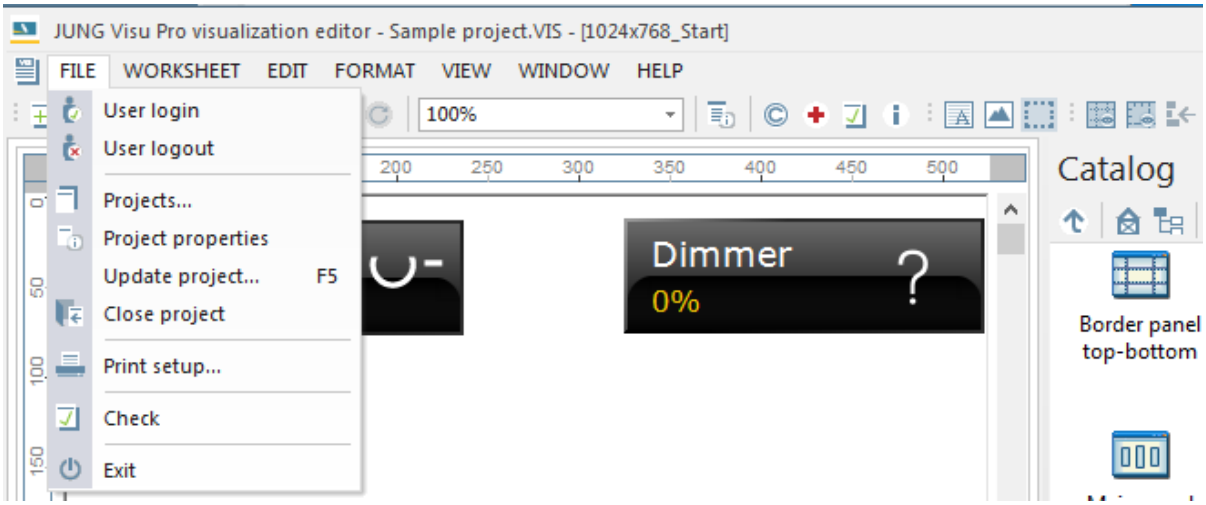
3 User Interface

The user interface of the visualization editor comprises the following main components:

- **The menu**
- **The toolbar**: The toolbar contains buttons which provide quick access to certain functions.
- **Workspace**: In the workspace you can load and edit worksheets.
- **Tool windows**: The tool windows contain important functions for the configuration of worksheets.
- **Message window**: The message window displays check results and system messages.

3.1 Menu

Menu item *File*



User Login: Opens a dialog to log in a user to the visualization system.

User Logout: Logs current user off the system.

Projects: Opens the dialog ***Project Organizer*** where you can create new projects and open, delete and rename existing ones.

Project Properties: Opens a dialog to define or update general project properties.

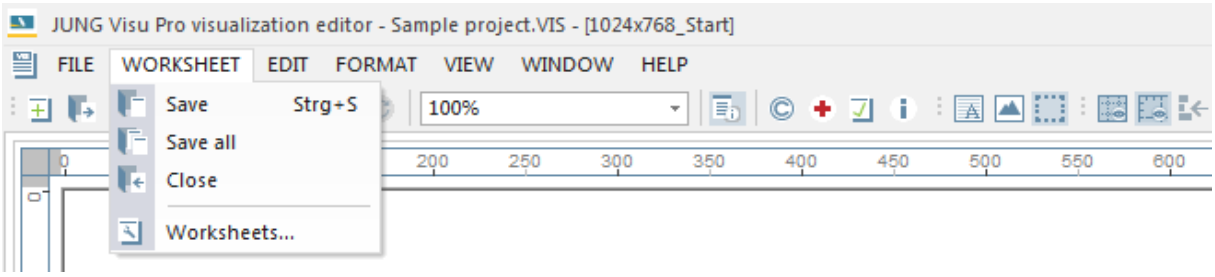
Close Project: Closes current project.

Print Setup..: Selection of printer and setting of printer properties.

Check: Opens a dialog to check project properties.

Exit: Quits the visualization editor. Shortcut ALT+F4.

Menu item *Worksheet*

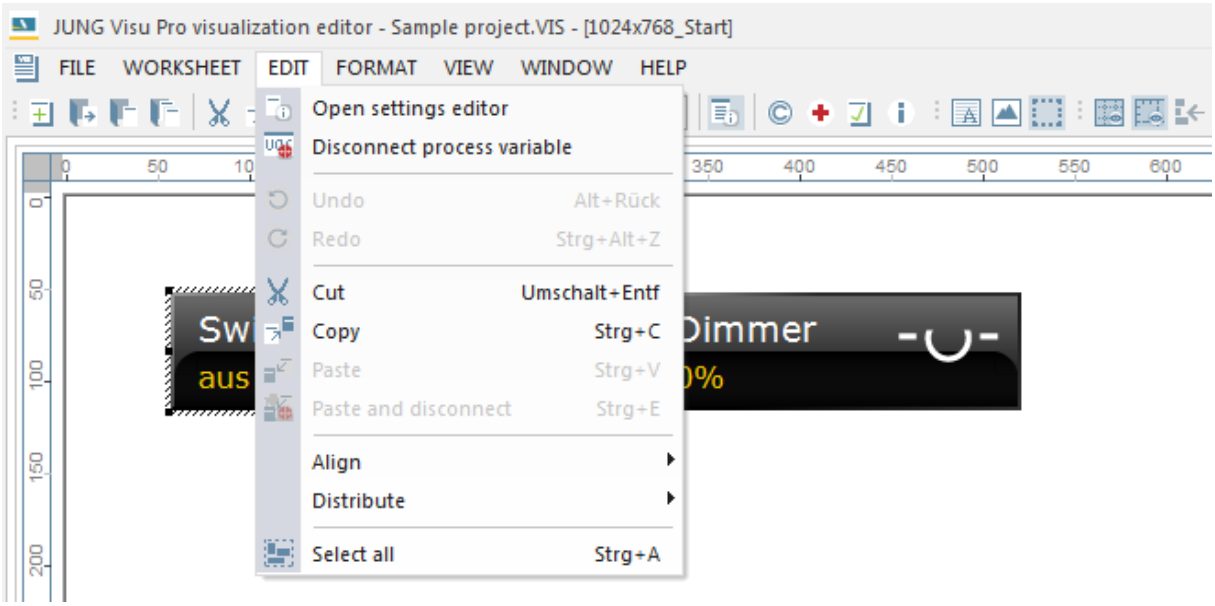


Save /Save All: Saves current worksheet / Saves all currently opened worksheets.

Close: Closes current worksheet.

Worksheets: Opens the dialog ***Worksheets*** where you can create new worksheets and organize existing ones.

Menu item *Edit*



Open settings editor: Opens the properties dialog of the selected element.

Disconnect Process Variable: Disconnects the link between the selected display element and the connected process variable. ***Multiple selection is possible***.

Undo: Reverses the last change carried out in the worksheet.

Redo: Reverses the undo command.

Cut: Cuts out the selected items and copies the content to the clipboard.

Copy: A display element can be copied and pasted to the same or a different worksheet.

Paste: Pastes items from the clipboard to the worksheet. The links between display elements and process variables remain unchanged. If the display element is inserted in the same worksheet, its name will be changed. If it is inserted in a different worksheet, its name remains unchanged. The name of a display element can exist only once in a worksheet. **Multiple selection possible**.

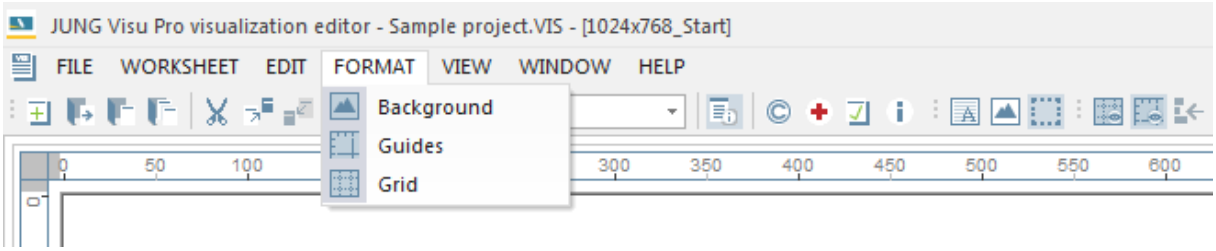
Paste and Disconnect: Pastes elements from the clipboard to a worksheet. The links between display element and process variables will be disconnected. If the display elements are inserted in the same worksheet, their names will be changed. If they are inserted in a different worksheet, their names remain unchanged. The name of a display element can exist only once in a worksheet. **Multiple selection possible**.

Align: It is possible to align several selected display elements, with these functions. For this, the edge of the outermost element is always used, at which the other elements orientate. The options are: Down, Up, Left and Right.

Distribute: Aligns several selected control elements in a constant distance from each other. It is possible to arrange them either horizontally or vertically.

Select All: Selects all items in the current plane.

Menu item *Format*

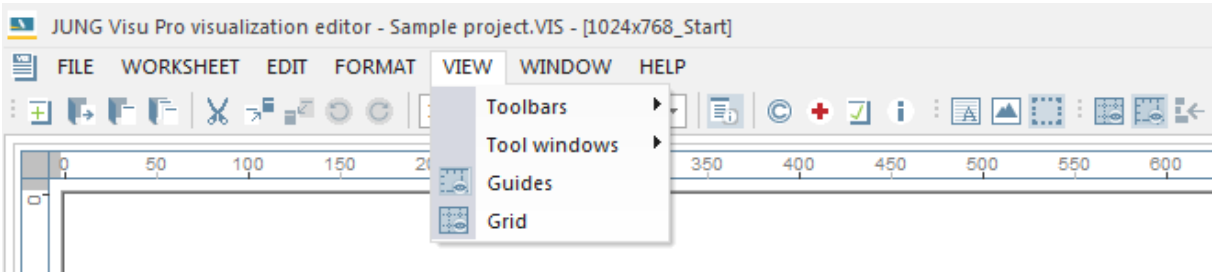


Background: Opens a dialog to define the background properties of the current worksheet.

Guides: Opens a dialog to edit guides.

Grid: Opens a dialog to define grid properties. The horizontal and vertical distances of the grid may be defined here. Optionally, the snap-to-grid function can be activated.

Menu item *View*



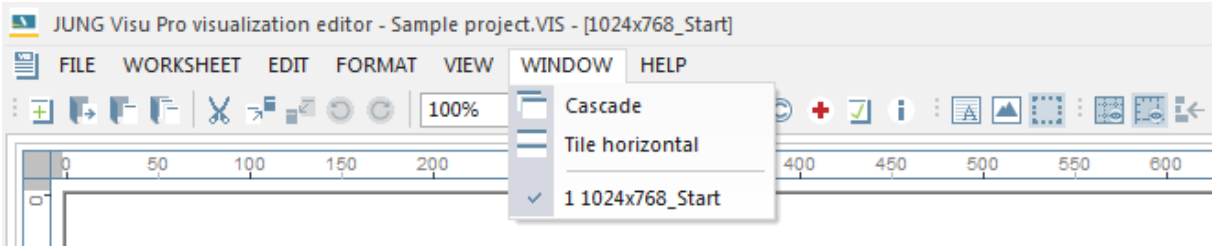
Toolbars: Used to toggle the visibility of toolbar elements.

Tool windows: Used to toggle the visibility of the tool windows.

Guides: Toggles the visibility of guides.

Grid*:* Toggles the visibility of the grid.

Menu item *Window*

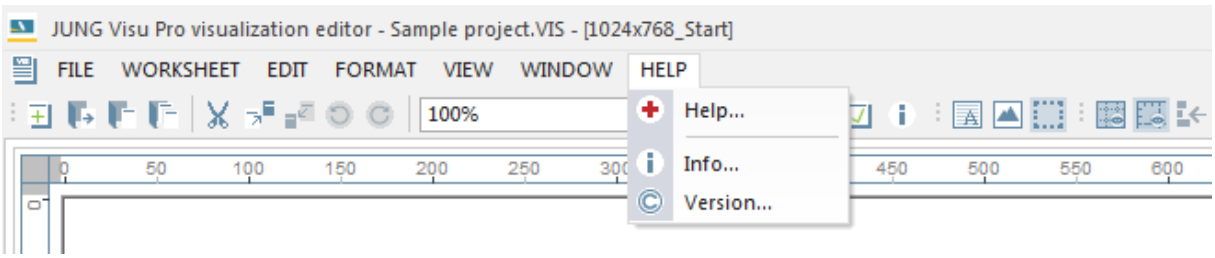


Cascade: The worksheet windows overlap.

Tile horizontal: Arranges windows as non-overlapping horizontal tiles.

The menu item ***Window*** shows the worksheets loaded in the editor. If more than one worksheet is open, the active worksheet is marked by a tick.

Menu item *Help*

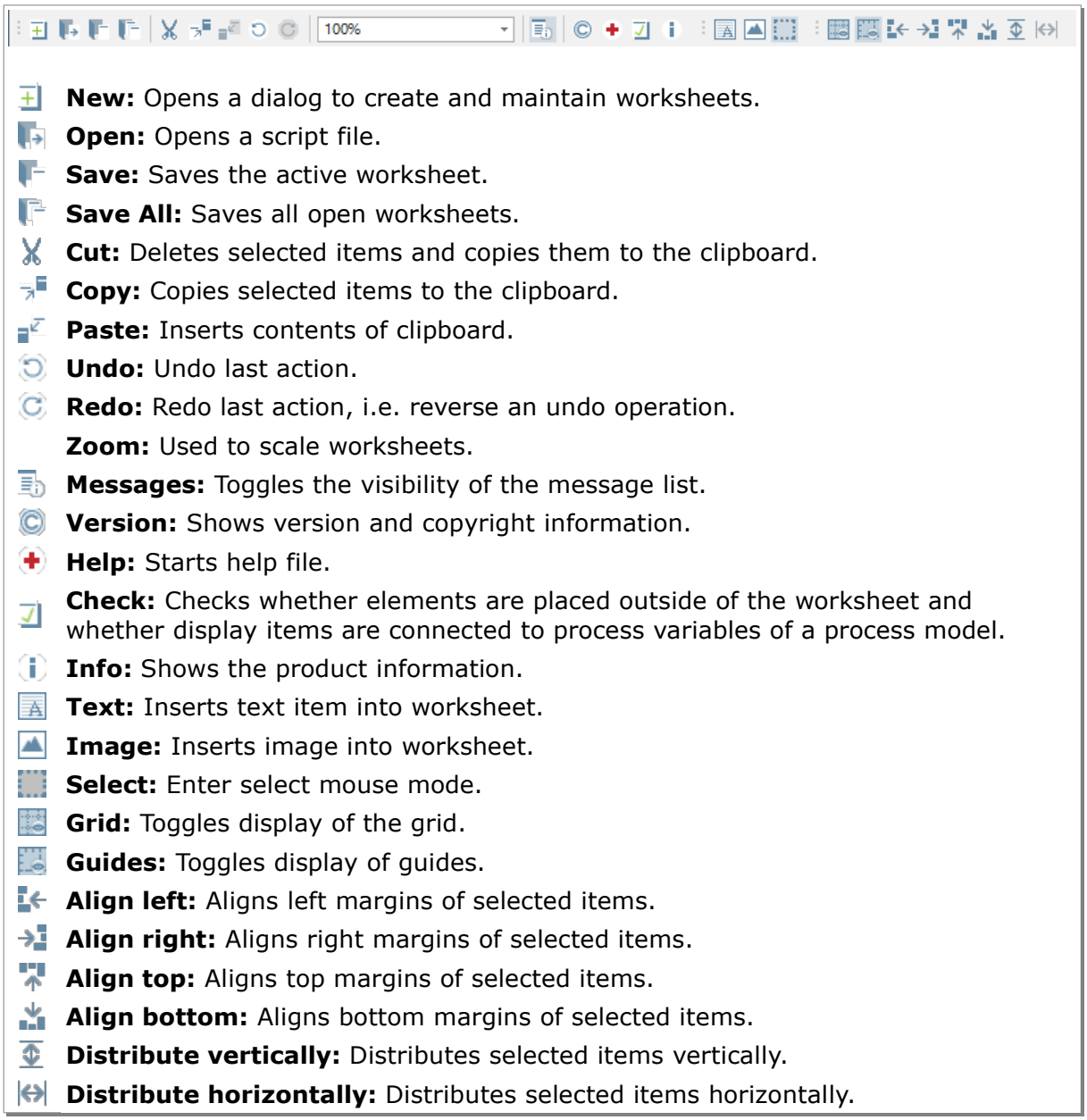


Help: Opens the help file. Shortcut: F1.

Info: Shows product and license information.

Version: Shows version and copyright information.

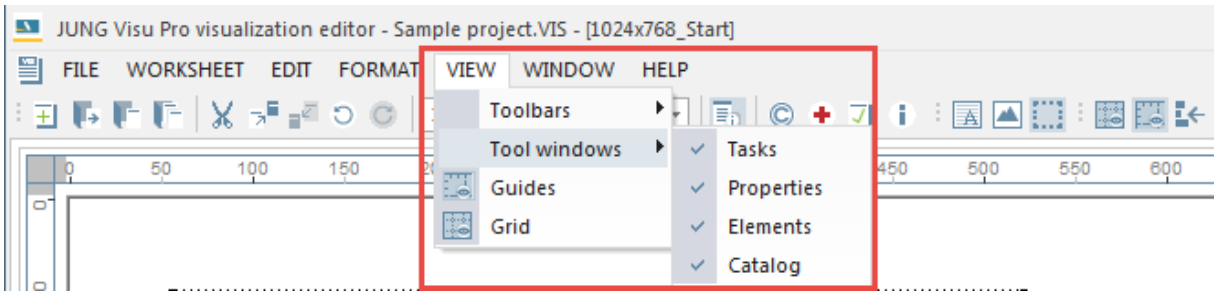
3.2 Toolbar



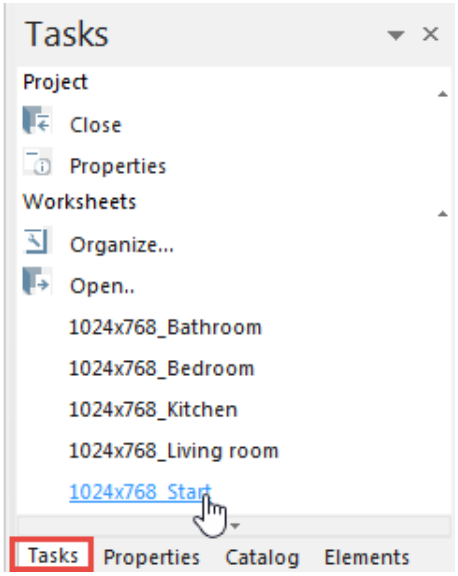
New: Opens a dialog to create and maintain worksheets.
Open: Opens a script file.
Save: Saves the active worksheet.
Save All: Saves all open worksheets.
Cut: Deletes selected items and copies them to the clipboard.
Copy: Copies selected items to the clipboard.
Paste: Inserts contents of clipboard.
Undo: Undo last action.
Redo: Redo last action, i.e. reverse an undo operation.
Zoom: Used to scale worksheets.
Messages: Toggles the visibility of the message list.
Version: Shows version and copyright information.
Help: Starts help file.
Check: Checks whether elements are placed outside of the worksheet and whether display items are connected to process variables of a process model.
Info: Shows the product information.
Text: Inserts text item into worksheet.
Image: Inserts image into worksheet.
Select: Enter select mouse mode.
Grid: Toggles display of the grid.
Guides: Toggles display of guides.
Align left: Aligns left margins of selected items.
Align right: Aligns right margins of selected items.
Align top: Aligns top margins of selected items.
Align bottom: Aligns bottom margins of selected items.
Distribute vertically: Distributes selected items vertically.
Distribute horizontally: Distributes selected items horizontally.

3.3 Tool Windows

The editor comprises tool windows which provide overviews, assistance and easy access to common tasks and recently used operations. According to the current situation, the content of the tool windows changes automatically.
To show or hide the tool windows, choose the menu option ***View*** and click ***Tool windows***.



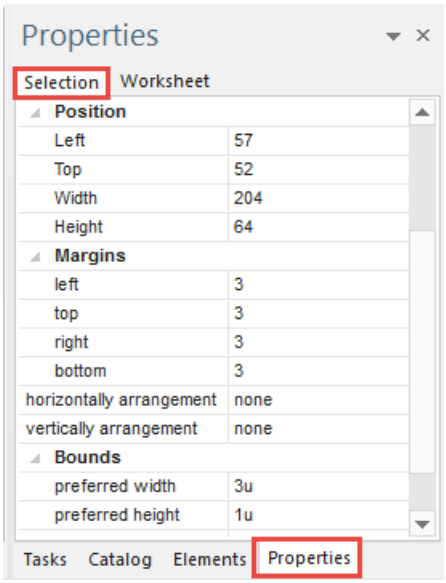
3.3.1 Tasks



The tool window ***Tasks*** shows frequently used tasks which can be selected with a single click.

3.3.2 Properties

The tool window ***Properties*** comprises switchable tabs: the tab ***Selection*** and the tab ***Worksheet***.



Selection

The tab ***Selection*** shows properties of items selected in the worksheet. The item properties can be changed directly in this window.

Name: The element name is an important specification which facilitates the editing of projects. It serves the internal processing of visualization projects.
During the editing process of a project, the number of elements contained in a worksheet increases. Therefore it is important to assign meaningful names to elements (panels or display elements) in order to have a better overview over them.

Type: Information about the element type.

Plane: Indicates the plane in which the selected element is positioned.

ID: Unique ID of the element.

Visibility: Toggles the visibility of the element. You can also set this option by selecting the checkbox under the icon in the tool window ***Elements***.

Group: You can close a panel to prevent the elements from being selected when clicked. Thus, only the closed panel can be selected, but not the panels or display elements contained in it.

You can also set this option by selecting the checkbox under the icon in the tool window ***Elements***.

Position

Left: Distance of the selected element from the left edge of the editor (in pixels).

Top: Distance of the selected element from the top edge of the editor (in pixels).

Width: Width of the selected element in pixels. This parameter can only be modified if the element is placed outside a panel. For elements placed within panels, the parameters **preferred width** and **preferred height** apply.

Height: Height of the selected element in pixels. This parameter can only be modified if the element is placed outside a panel. For elements placed within panels, the parameters **preferred width** and **preferred height** apply.

Margins

Here you can configure the width of the outer margin of the element.

Horizontal arrangement: Configuration of the horizontal arrangement of the element within its cell.

Vertical arrangement: Configuration of the vertical arrangement of the element within its cell.

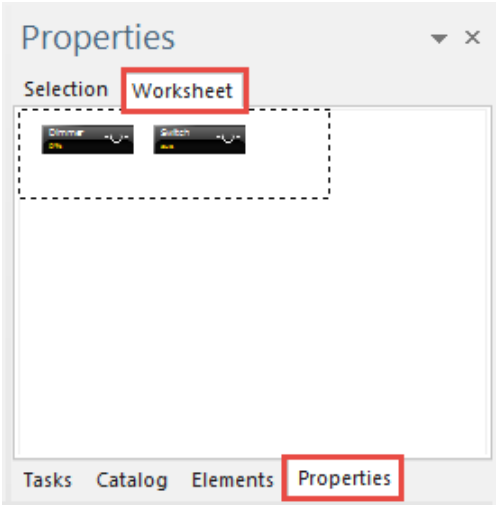
Bounds

Preferred width and height: The parameters *preferred width* and *preferred height* only apply to the size of the elements within a panel. They do not apply for elements placed outside of panels.

The preferred size can be edited. When placed in a panel, an element takes on the preferred size, provided the panel offers enough space.

The preferred size can be specified in pixels or in unit. 1 unit = 64 pixels.

Worksheet



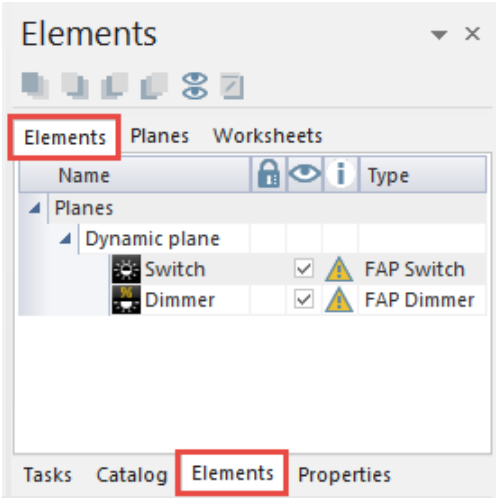
The tab ***Worksheet*** shows a reduced view of the active worksheet.

The visible window content is marked by a perforated frame. You can scroll in the worksheet by shifting this frame: press and hold down the left mouse button and drag the frame to the desired position.

3.3.3 Elements

The tool window ***Elements*** comprises three tabs whose content depends on the current situation in the editor.

Elements



This tab shows all elements contained in the active plane. It offers the following functions:

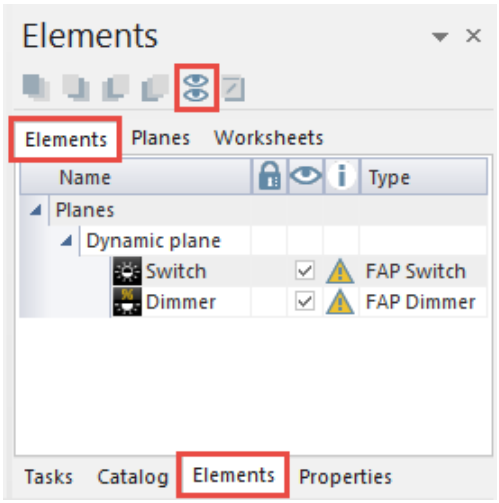
Select elements: Click on an element in the ***Elements*** tab to select it in the worksheet (multiple selection possible).

Toggle visibility of an element: By selecting the checkbox under the icon elements can be hidden. To show an element, deselect the checkbox.

Close panel: You can close a panel to prevent the elements from being selected when clicked. Thus, only the closed panel can be selected, but not the panels or display elements contained in it. To close a panel, select the corresponding checkbox under the icon .

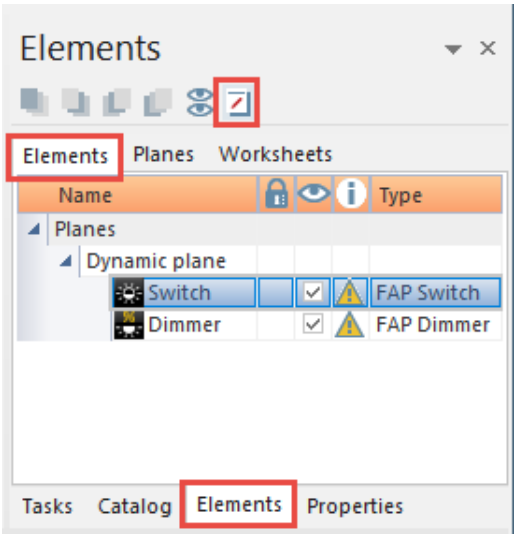
Grade elements: You can rearrange the position of elements relative to each other using the buttons in the toolbar of the Elements window:

Synchronize view: With the help of the synchronize button or by pressing the F3 key, the window is shifted to display the element selected in the Elements tool window.



If the synchronize button remains activated, the window content of the worksheet is shifted automatically to the element selected in the Elements tool window.

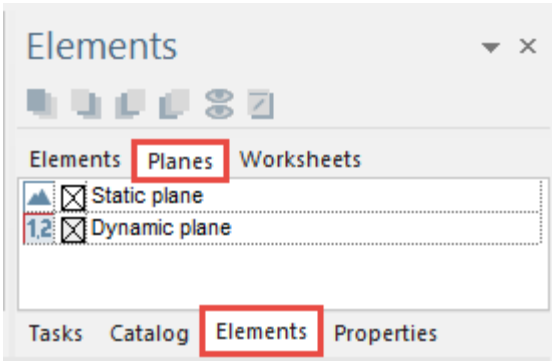
Edit element: By clicking the button or pressing the F4 key, the properties dialog opens where the selected element can be edited.



The symbol ⚠ indicates that the display element is not connected to a process variable or that the element is placed out of the worksheet.

Planes

The worksheet comprises two planes which may be activated or deactivated. One plane is always active and editing commands relate to this plane.



The active plane can be selected in this window.

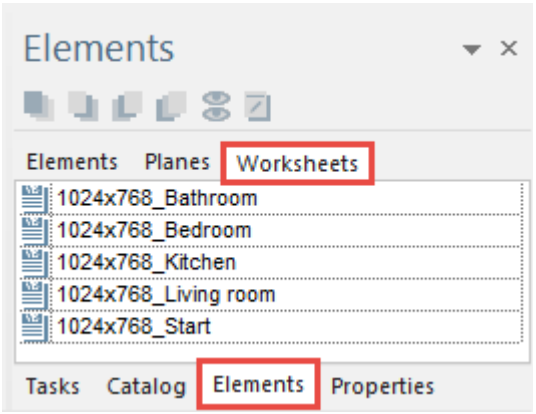
Alternatively, the active plane can be selected in the context menu of the worksheet: open the context menu of the worksheet with a right click on an empty space in the worksheet and choose the desired plane.

Static plane: Used for image objects.

Dynamic plane: Used for dynamic items. All display items for process variables, for instance, must be inserted into this plane.

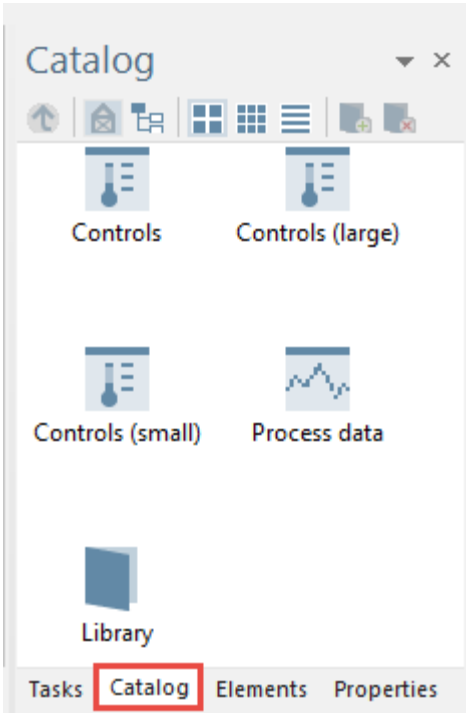
Worksheets

This tab shows a list of all worksheets created in the visualization project. To open a worksheet, double-click its name.



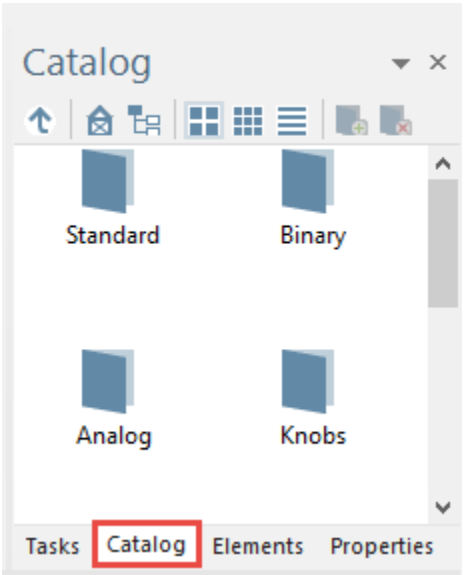
3.3.4 Catalog

The Catalog window contains all items which can be used when configuring the visualization project.



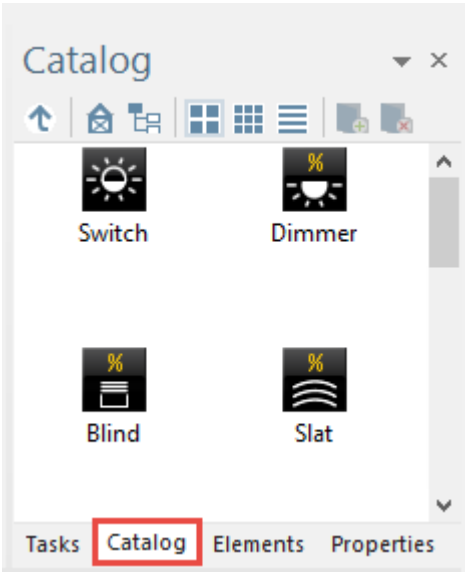
Catalog

Overview of all object groups.

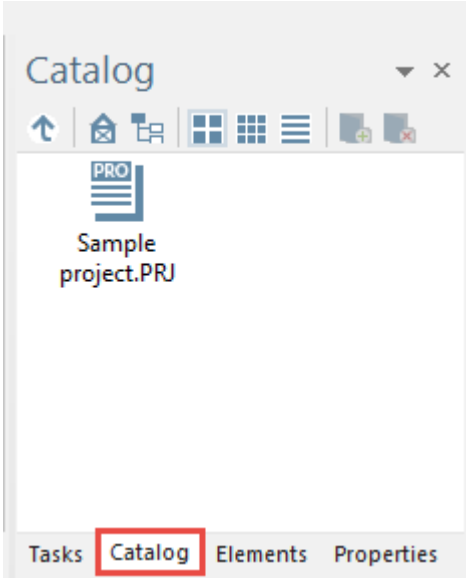


Controls

Overview over all usable elements which can be used on a worksheet.

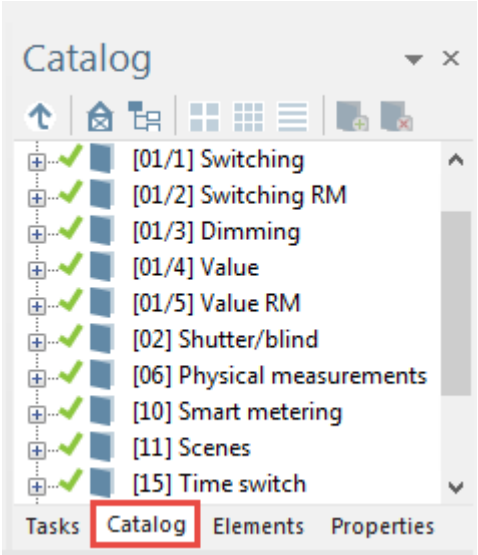


To insert a parameter display into a worksheet, press and hold down the left mouse button and drag the parameter display into the worksheet window. Release the mouse button to drop the item at the cursor position.



Process models

Contains the connected process model.



Process variables

To connect a display element with a process variable, press and hold down the left mouse button on the process variable and drag it into the display element. If the type of the process variable matches the display element, you can drop the dragged variable.

3.4 Window Management

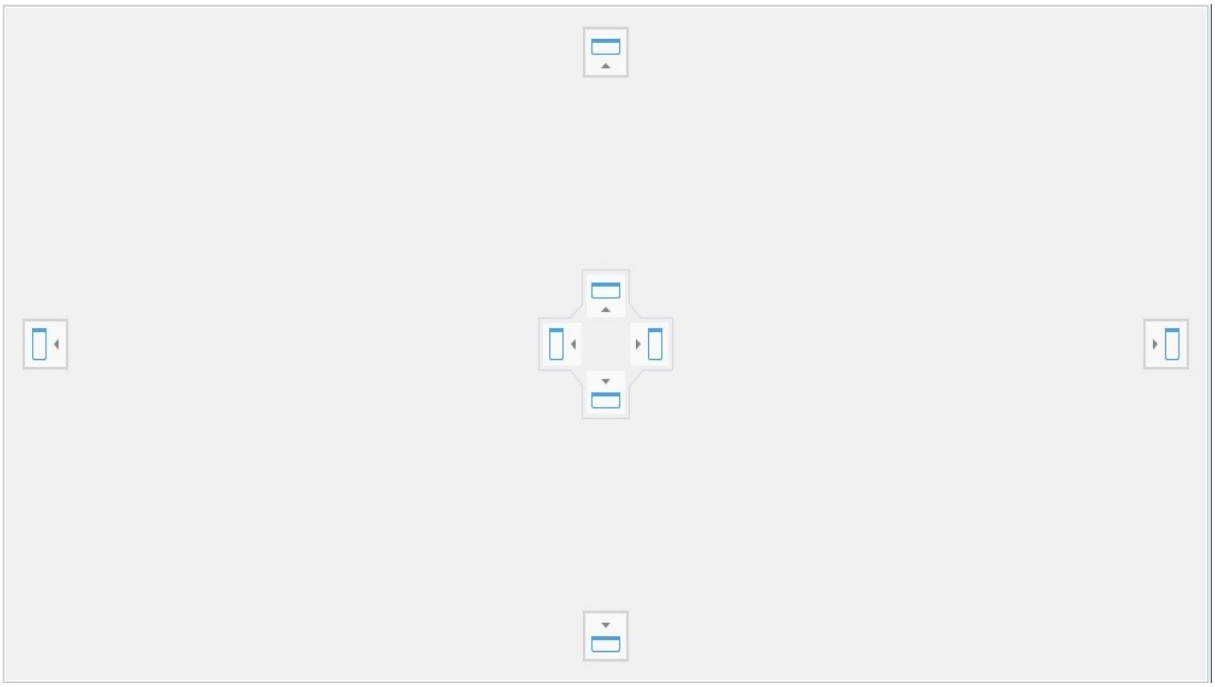
You can undock tool windows and the message window from their positions and arrange them anywhere in the editor or dock them to predefined positions.

Floating windows

Click the title bar of the window you wish to rearrange and drag it to the appropriate position in the editor. The window exists now as a separate floating window.

Docked windows

With the help of **docking guides** you can dock windows to predefined positions in the editor. The docking guides are displayed when you start dragging a window from its position. They indicate where the window may be docked. The number of the possible docking positions depends on the target window and on the window you want to dock.

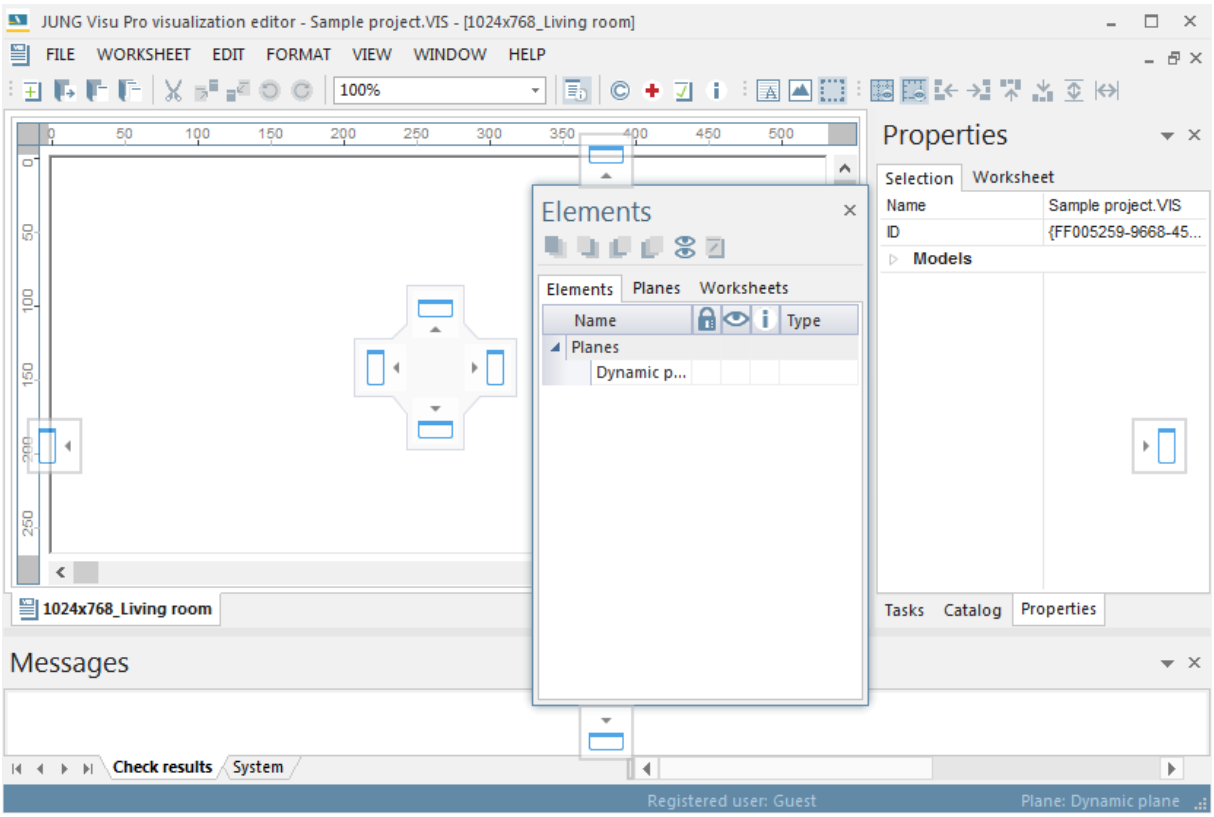


There are two options for docking a window:

- **relatively to the entire editor:** for this purpose, docking guides are displayed at the top, bottom, left and right side of the editor edge, indicating the possible docking positions in the entire editor area.
- **relatively to an existing window:** for this purpose, docking elements are displayed in the center of the window the mouse cursor is currently over, indicating the possible docking positions within this window.

How to dock a window:

- Click the window's title bar with the left mouse button, hold down the mouse button and start dragging the window to the area you wish to dock it to.
- When you start dragging the window, docking guides are displayed indicating the possible docking positions within the target window.
- When you move the mouse cursor over one of the docking guides a semi-transparent blue rectangle is displayed indicating the dropping area. This is the area the window will be docked to if you release the mouse button.



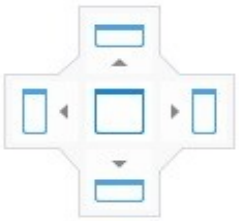
The picture above shows the docking guides in the editor.

- To position the window in the highlighted area, release the mouse button.

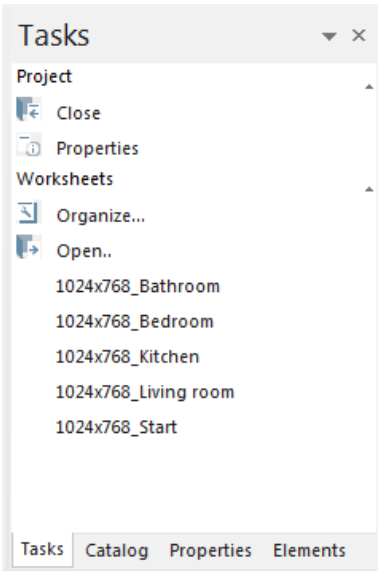
Tabbed windows

Tool windows can be docked to each other to form tabbed groups. To insert a window as a tab page within an existing window:

- Click the window's title bar, hold down the mouse button and drag the window to the target window.
- During the drag operation docking guides are displayed indicating the possible docking positions within the target window. The center docking guide represents a tabbed group.



- Release the mouse button over the central docking guide to insert the window as a new tab page into the existing window.

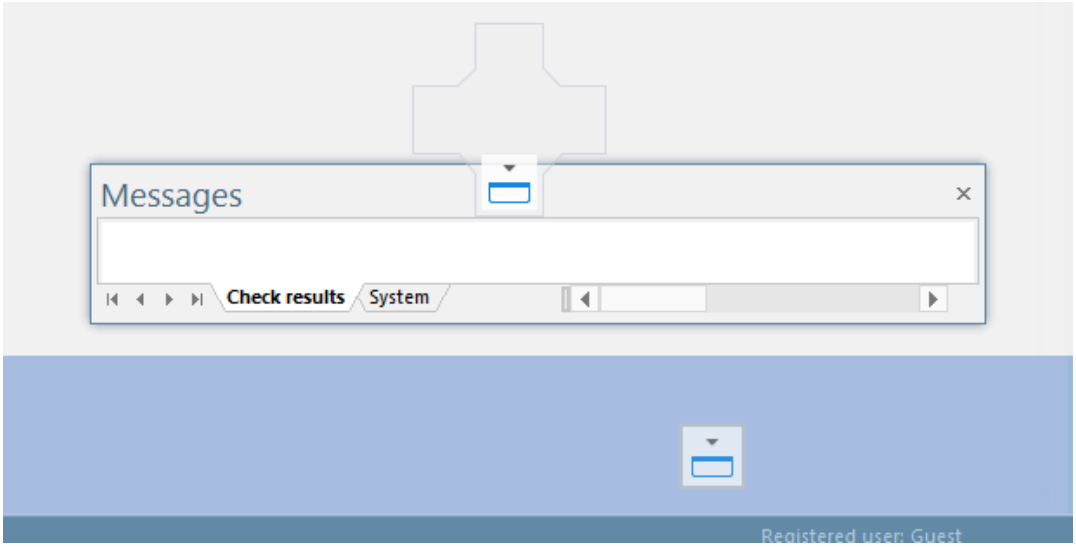


The tool windows docked in the tab group are displayed as tabs in the tab bar at the bottom of the window.

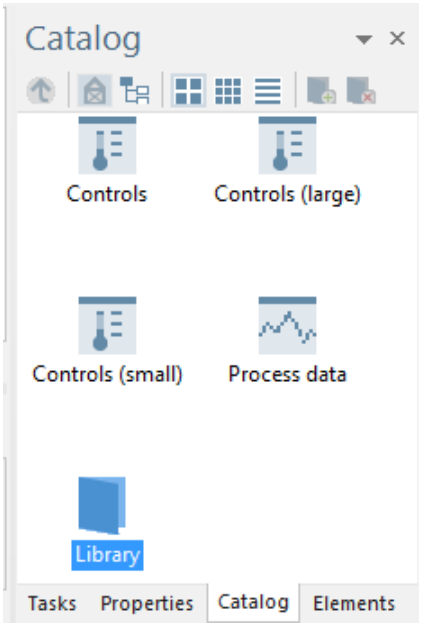
To remove a window from its arrangement as tabbed window, click its tab and drag it out of the tabbed group.

Docking the message list

The message list may be docked to the bottom of the workspace edge or to the bottom of the editor edge.



3.5 The Library



The library is a folder in which you can save predefined panels or single display elements (modules) as templates which can be reused in different worksheets. A predefined panel may be a single panel or a panel construction containing several panels. Complete worksheets **cannot** be saved in the library.

The library is located in the tool window ***Catalog***.

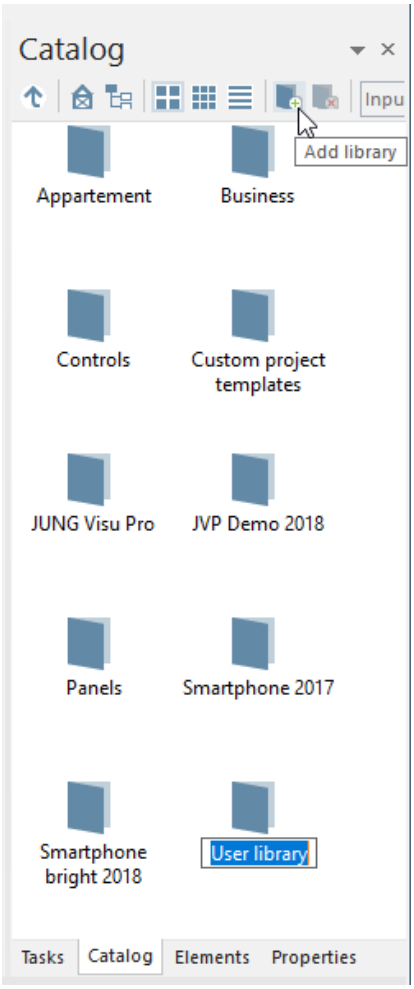
Usage of the library

The library plays an important role when editing worksheets, since the modules saved in it can be reused as often as required. Thus, certain components can be used throughout all worksheets of a project.

That way the worksheets are provided with a consistent structure. Moreover, the reuse of modules reduces the time required for the configuration of a visualization project in a significant way.

Create a user library

The library contains folders with predefined modules. To create a new user library, press the Insert key. A folder with the default name ***User library*** is created.



To change the name of the folder, select the folder and click its name. The input field opens where you can enter a new name. You can change the name also by pressing the **F2 key**.

Save a module to the library

To save a module to a folder in the library, select the module in the worksheet by pressing the left mouse button, hold down the mouse button and drag the module to the library. Connections to process variables will not be saved.

Use a module from the library

To insert a module from the library into the active worksheet, select the module by pressing the left mouse button, hold down the mouse button and drag the module to the worksheet. With pressed alt key, the element is inserted at the place it was as it was copied to the library.

Predefined modules

The folders Facility Pilot and Smartphone contain predefined modules for your projects. The other folders can be used to save your own modules.

JUNG Visu Pro: You can not save new modules in this folder. It contains modules for creating a weather page.

Smartphone: You can add new modules in this folder. It contains examples for worksheets that are optimized for smartphones.

4 Functions

4.1 Project Organizer

In the dialog ***Project Organizer*** you can create new visualization projects and open, delete and rename existing projects.

To open the Project Organizer, choose the menu item ***File*** and click ***Projects***. Alternatively, choose the tool window ***Tasks*** and click under ***Projects*** on ***Organize***.

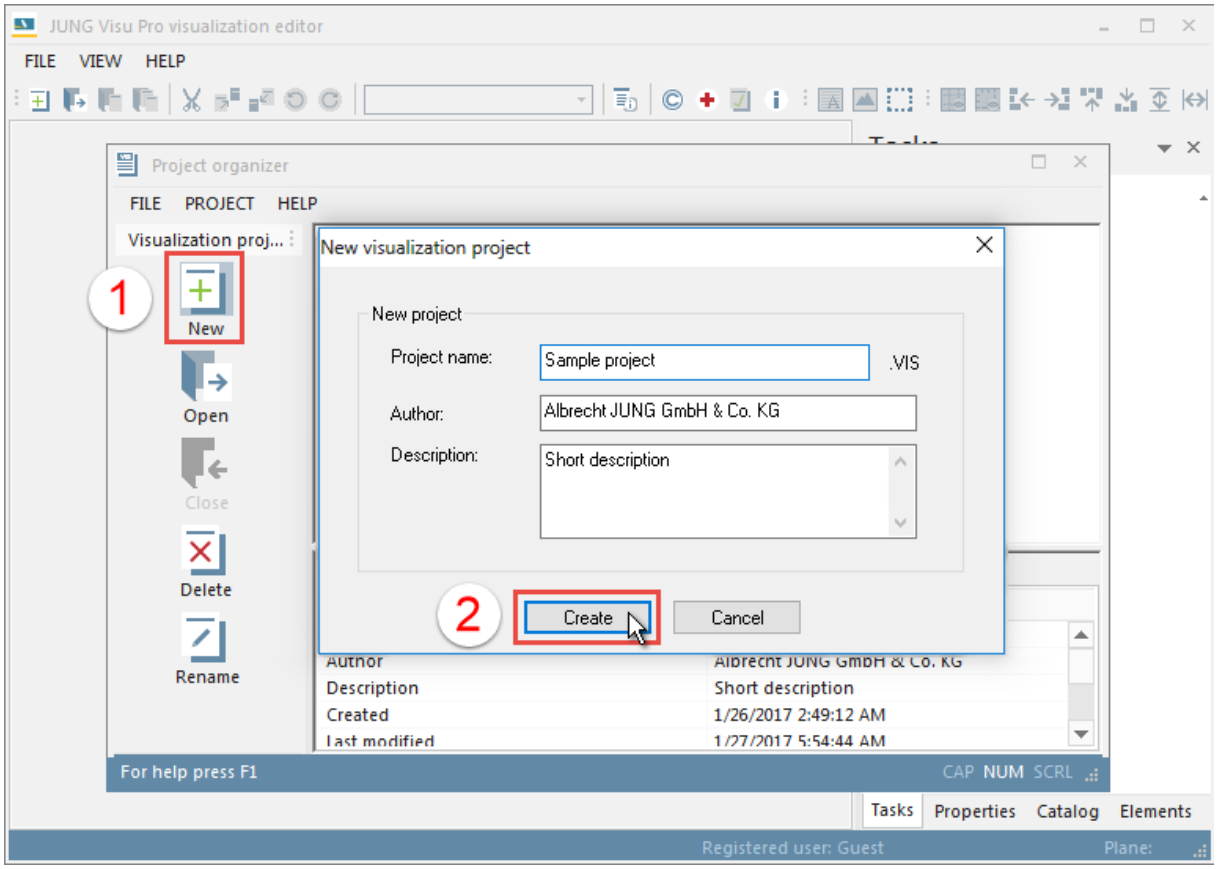
In the Project Organizer you can use the buttons in the sidebar or the commands in the menu item ***Project***. The Project Organizer offers the following functions:

- **New:** Create a new visualization project.
- **Open:** Load a project into the visualization editor.
- **Close:** Closes the current project.
- **Delete:** Delete the selected project.
- **Rename:** Rename the selected project.

Create project:

To create a new project, choose in the visualization editor the menu item ***File*** and click ***Projects***. Alternatively, choose the tool window ***Tasks*** and click under ***Projects*** on ***Organize***. The dialog ***Project Organizer*** opens.

Click the button ***New***. The dialog ***New Visualization Project*** opens.



Enter the data of the new project:

Project name: Name of the visualization project.

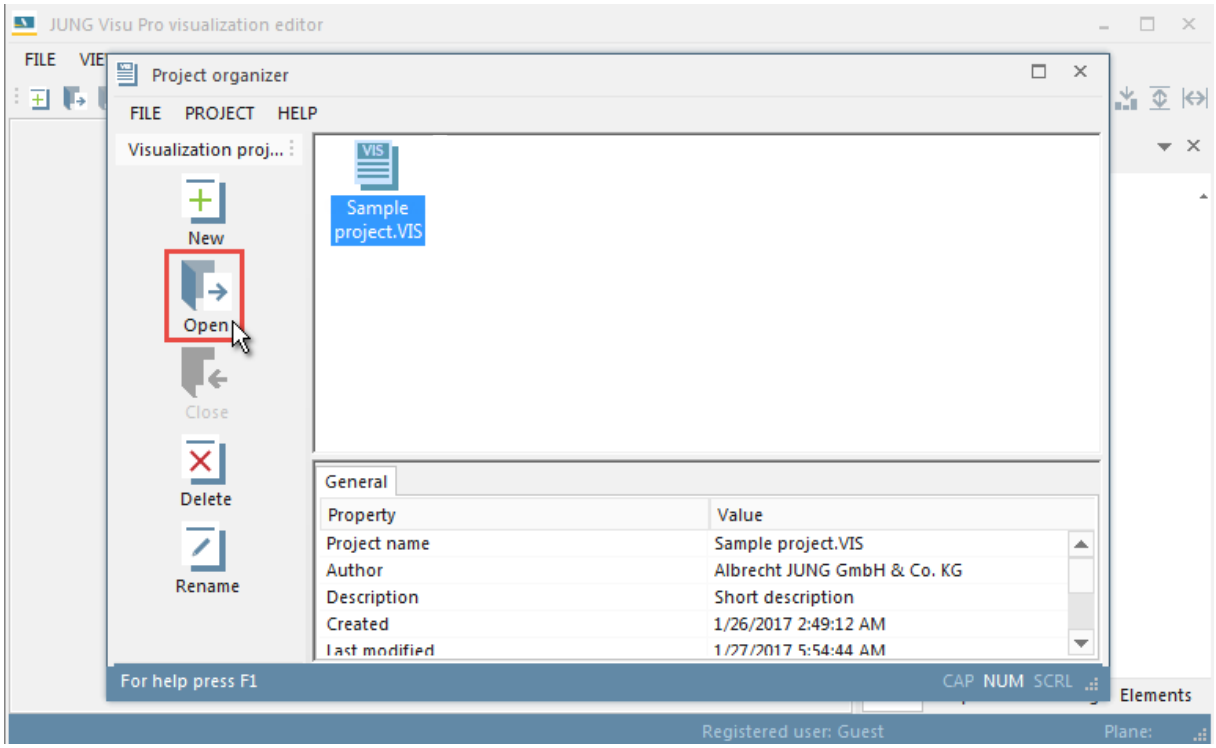
Author and description: These entries are useful but not mandatory. They can be entered or changed later.

After entering the data click the button ***Create***. Now the project is available in the Project Organizer and can be started from here.

Open project:

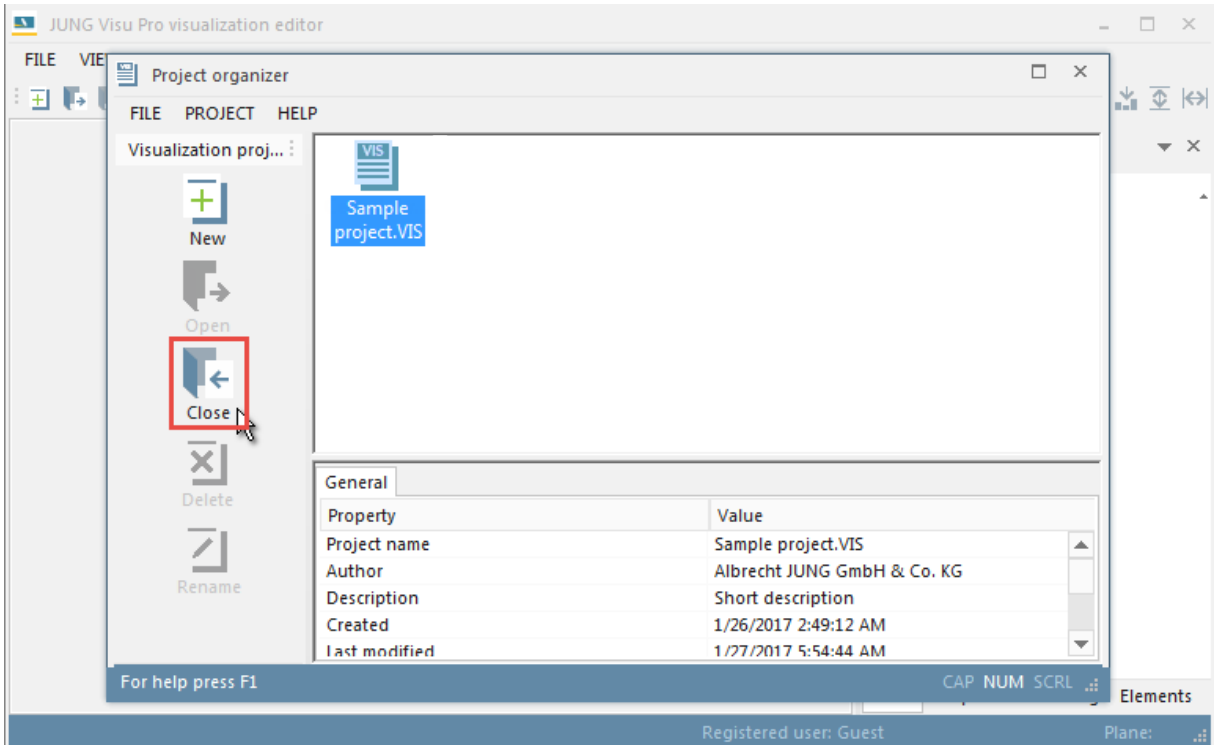
To load the project into the visualization editor,

- double-click the project icon or
- select the project and click the button ***Open*** in the sidebar or
- select the project, choose the menu item ***Project*** and click ***Open***.



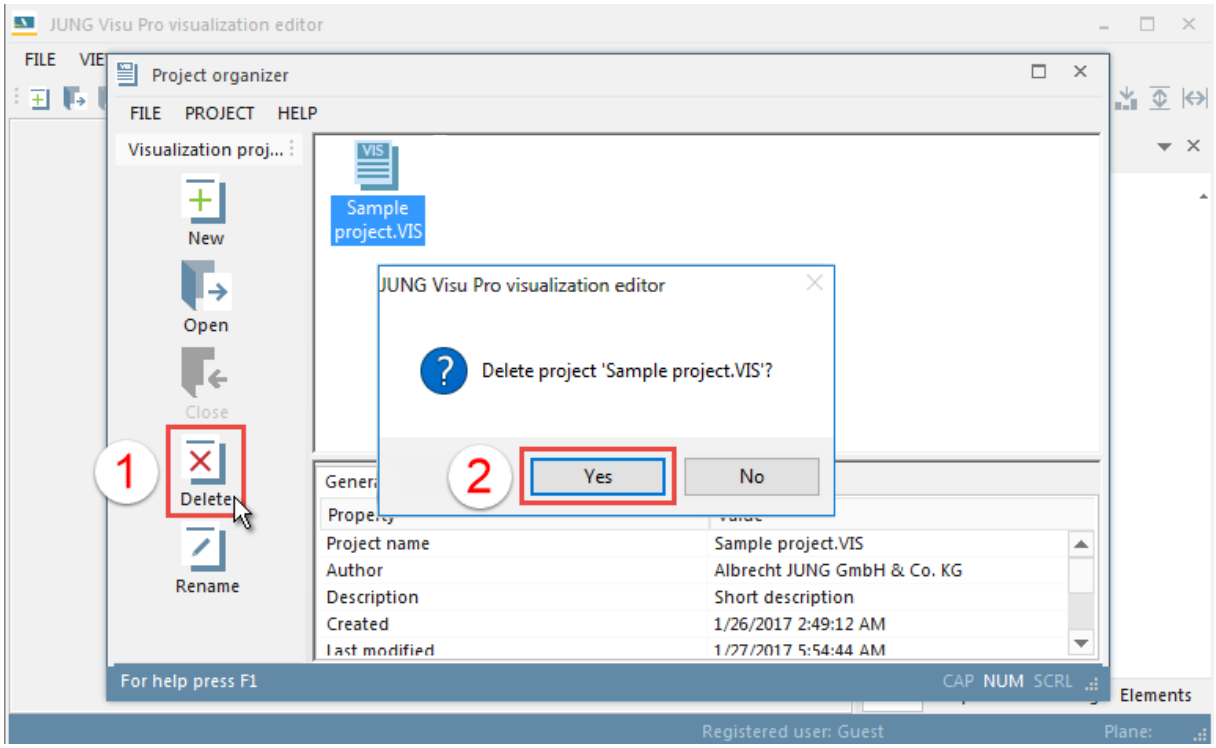
Close project:

To close the current project, select the project and click in the sidebar the button ***Close***. Alternatively, choose the menu item ***Project*** and click ***Close***.



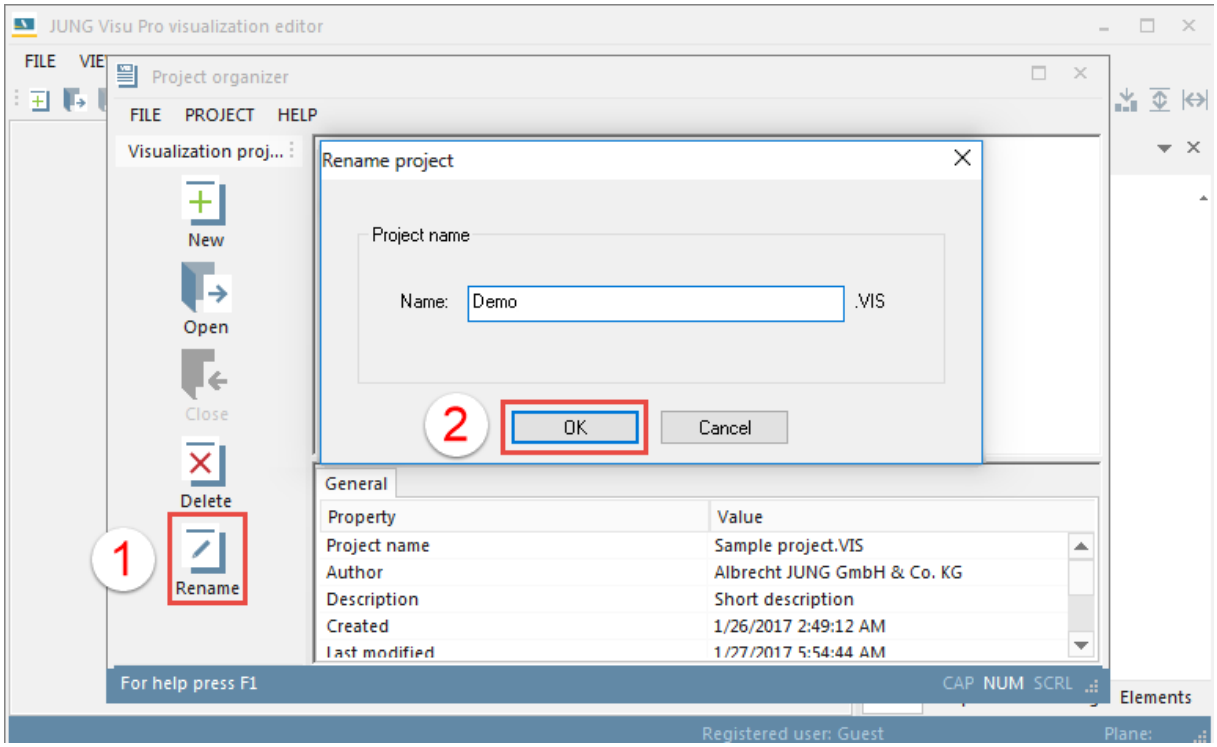
Delete project:

To delete a project, select the project and click in the sidebar the button ***Delete***. Alternatively, choose the menu item ***Project*** and click ***Delete***. Note that the project must be closed before you can delete it.



Rename project:

Note that a project must be closed before you can rename it. Select the project and click in the sidebar the button ***Rename***. Alternatively, choose the menu item ***Project*** and click ***Rename***. The dialog ***Rename project*** opens.



Enter the new project name and click ***OK*** to submit the change and to close the dialog. The extension ***.VIS*** is added automatically.

4.2 Project Properties

The dialog ***Project Properties*** contains important information about the project. It comprises the following tabs:

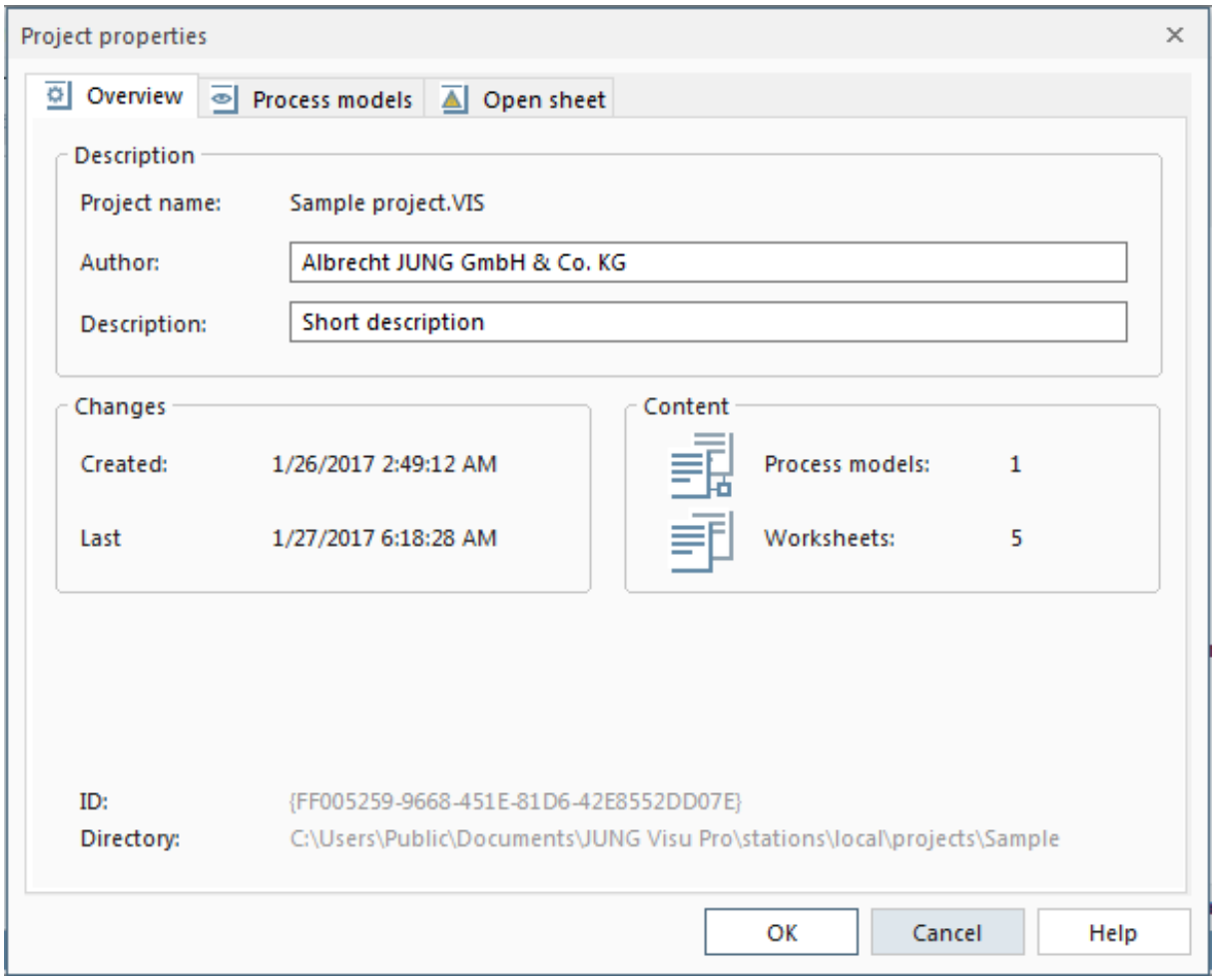
- **Overview**
- **Process models**
- **Open sheet**

To open the dialog, choose the menu item ***File*** and click ***Project Properties***.

Note that any changes made in the process model will only be committed to the visualization project if you close the dialog box ***Project properties*** by pressing ***OK***. If you close it by pressing ***Cancel***, the changes won't be committed. Hence, you may safely check the options of the properties window.

Tab *Overview*

The tab ***Overview*** shows general properties of the project loaded. Author and description of the project may be changed here.

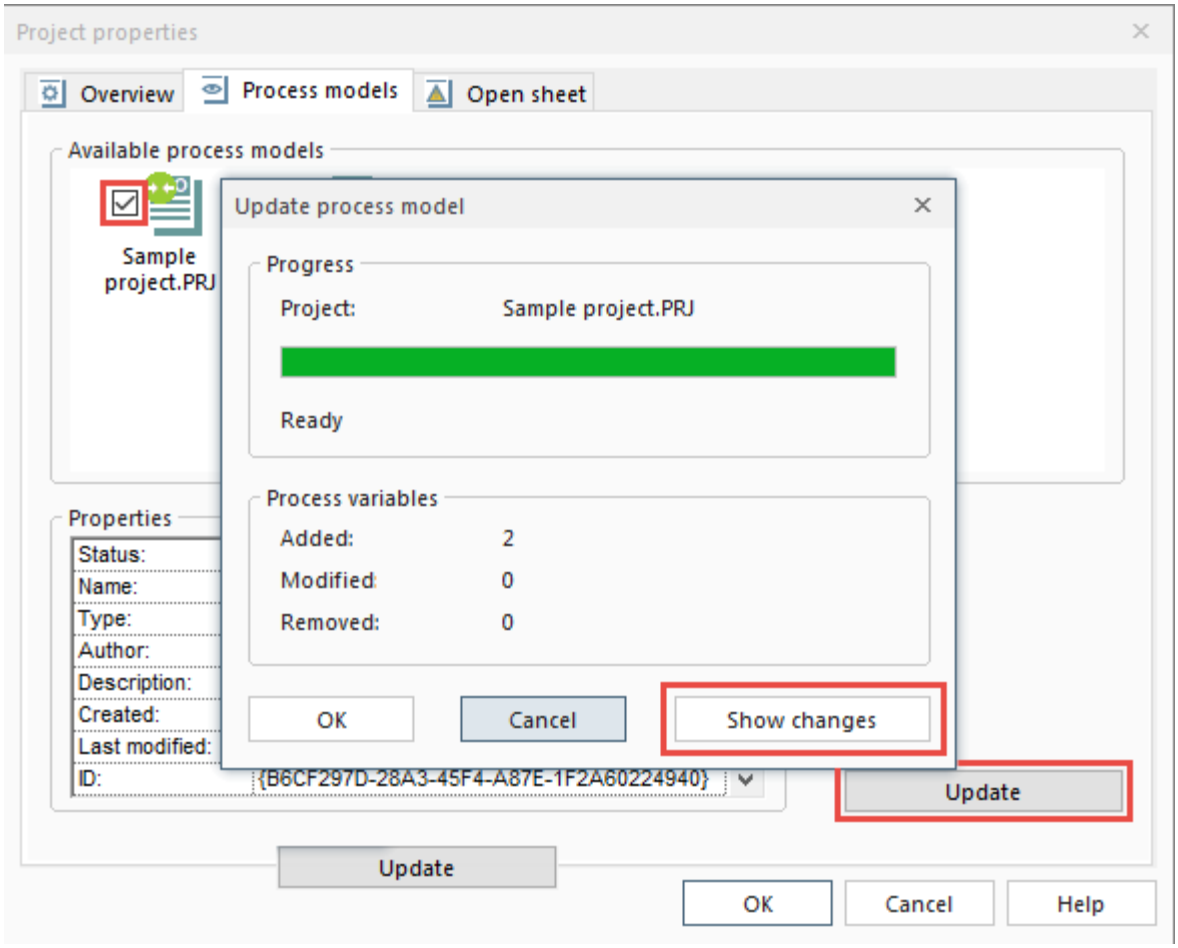


Tab *Process models*

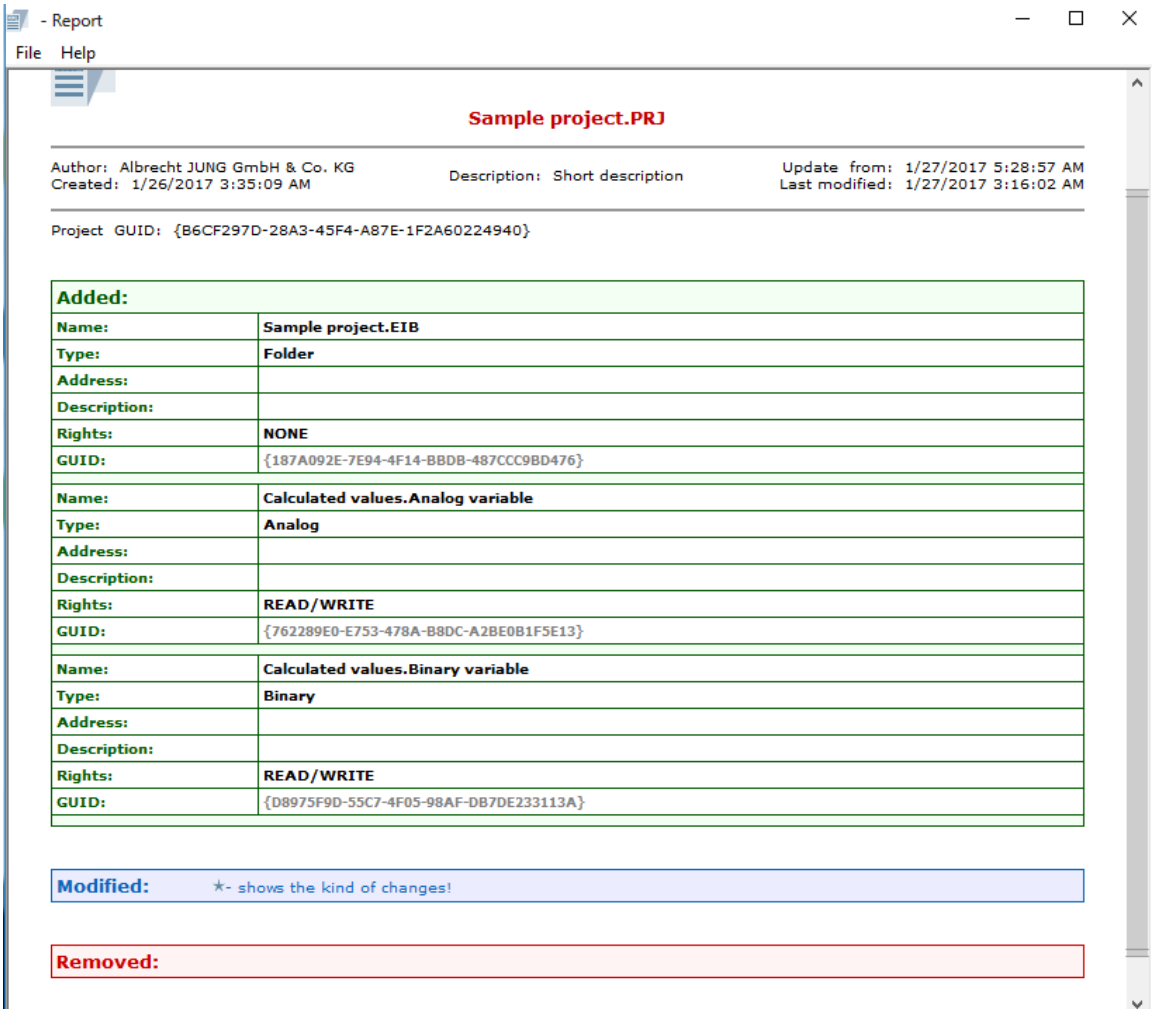
The tab ***Process models*** shows all process models available in the system. To connect a process model with the visualization project, select the checkbox in front of the corresponding project icon.

The dialog ***Update Process Model*** opens automatically to import the process variables of the process model.

If the process model has been modified, the button ***Show changes*** is enabled.

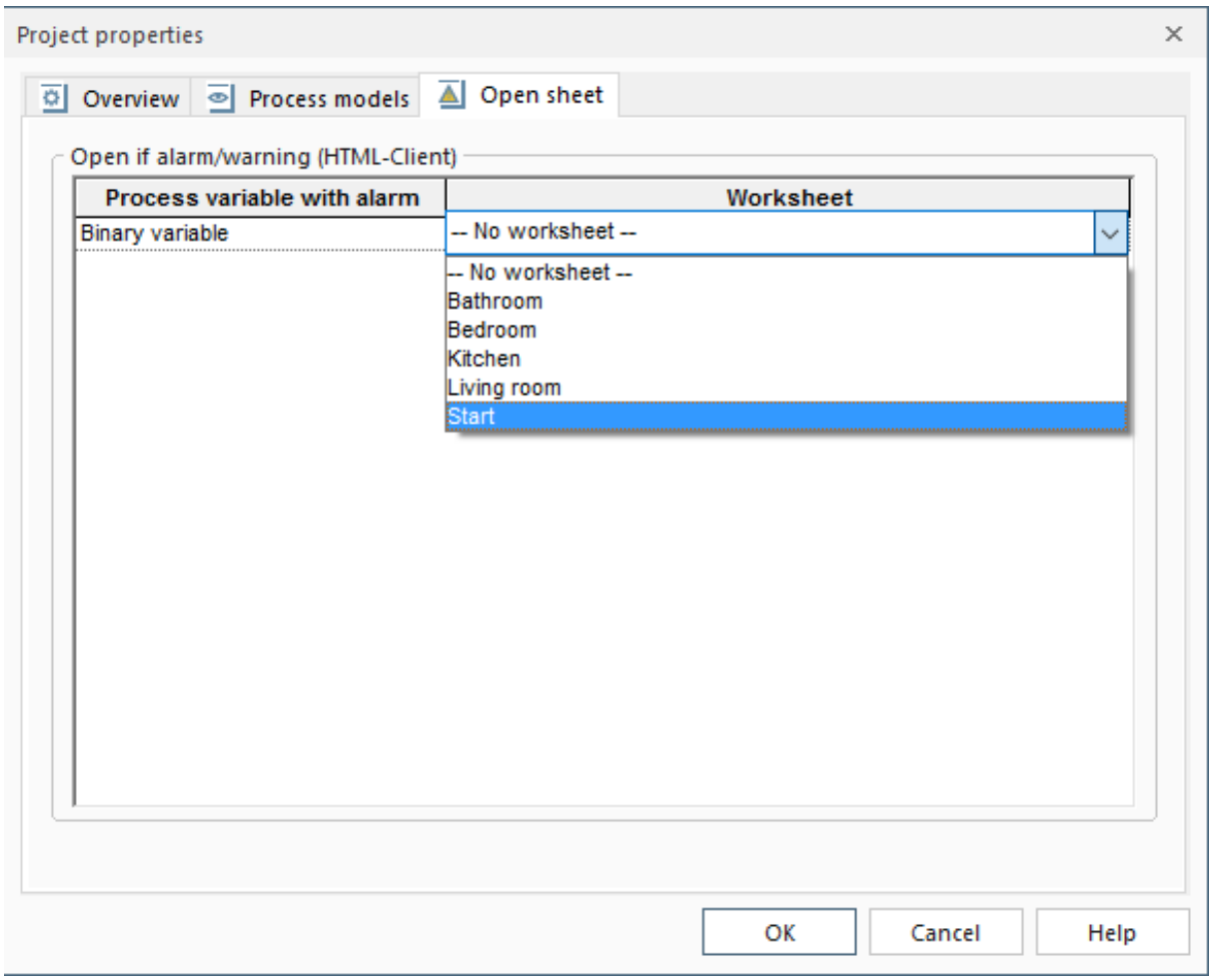


A click on the button ***Show changes*** opens a report containing detailed information about the update operation.


- Report
File Help
Sample project.PRJ
Author: Albrecht JUNG GmbH & Co. KG
Created: 1/26/2017 3:35:09 AM
Description: Short description
Update from: 1/27/2017 5:28:57 AM
Last modified: 1/27/2017 3:16:02 AM
Project GUID: {B6CF297D-28A3-45F4-A87E-1F2A60224940}
Added:
Name: Sample project.EIB
Type: Folder
Address:
Description:
Rights: NONE
GUID: {187A092E-7E94-4F14-BBDB-487CCC9BD476}
Name: Calculated values.Analog variable
Type: Analog
Address:
Description:
Rights: READ/WRITE
GUID: {762289E0-E753-478A-B8DC-A2BE0B1F5E13}
Name: Calculated values.Binary variable
Type: Binary
Address:
Description:
Rights: READ/WRITE
GUID: {D8975F9D-55C7-4F05-98AF-DB7DE233113A}
Modified: *- shows the kind of changes!
Removed:

Tab *Open sheet*

In the tab ***Open sheet*** you can specify the worksheet which opens, when an alarm is triggered.



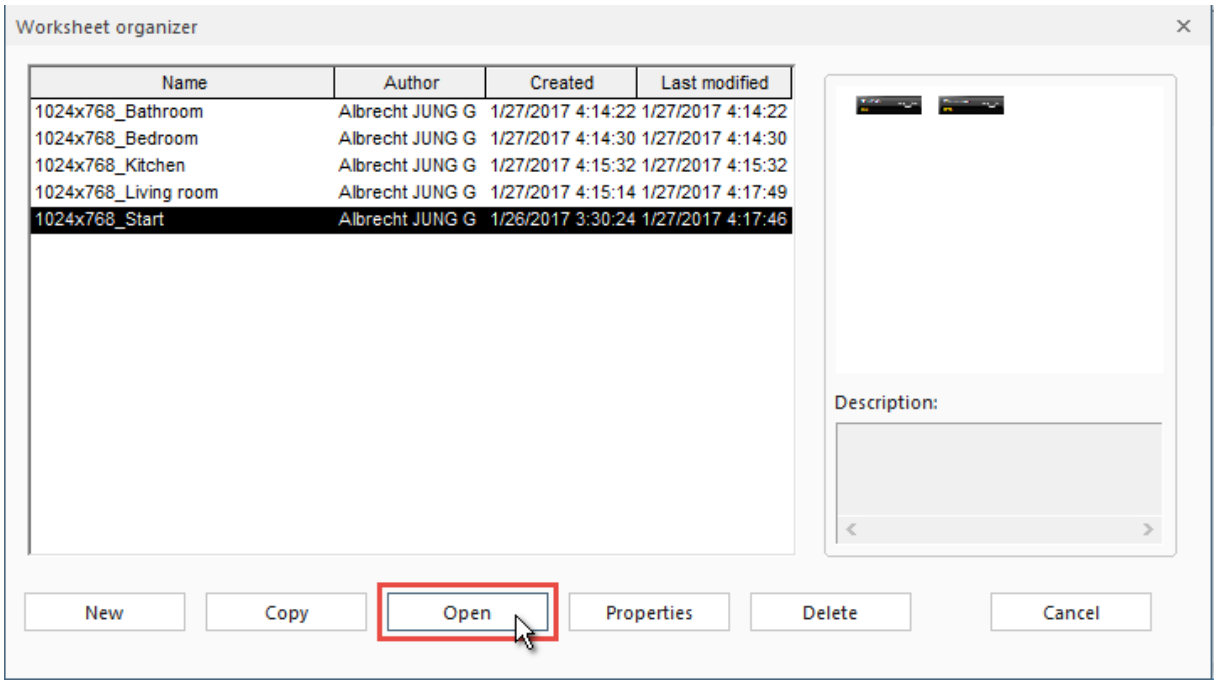
Click ***OK*** to close the dialog and to submit the changes. Otherwise press ***Cancel***.

Note: If settings have been made here, they can be deactivated for several users in the administration interface of the server. This has the effect that only certain registered users see the page views when they are triggered by alarms.

4.3 Worksheet Organizer

In the Worksheet Organizer you can create new worksheets as well as open and delete existing worksheets. Moreover, you can edit the properties of worksheets.

To open the dialog, choose the menu item ***Worksheet*** and click ***Worksheet Organizer***. Alternatively, choose the tool window ***Tasks*** and click under Worksheets on ***Organize***.



New: Create a new worksheet.

Copy: Copies the selected worksheet.

Open: Open the selected worksheet.

Properties: Opens the dialog ***Worksheet Properties*** where you can change the general property settings of the worksheet.

Delete: Deletes the selected worksheet.

Cancel: Closes the dialog.

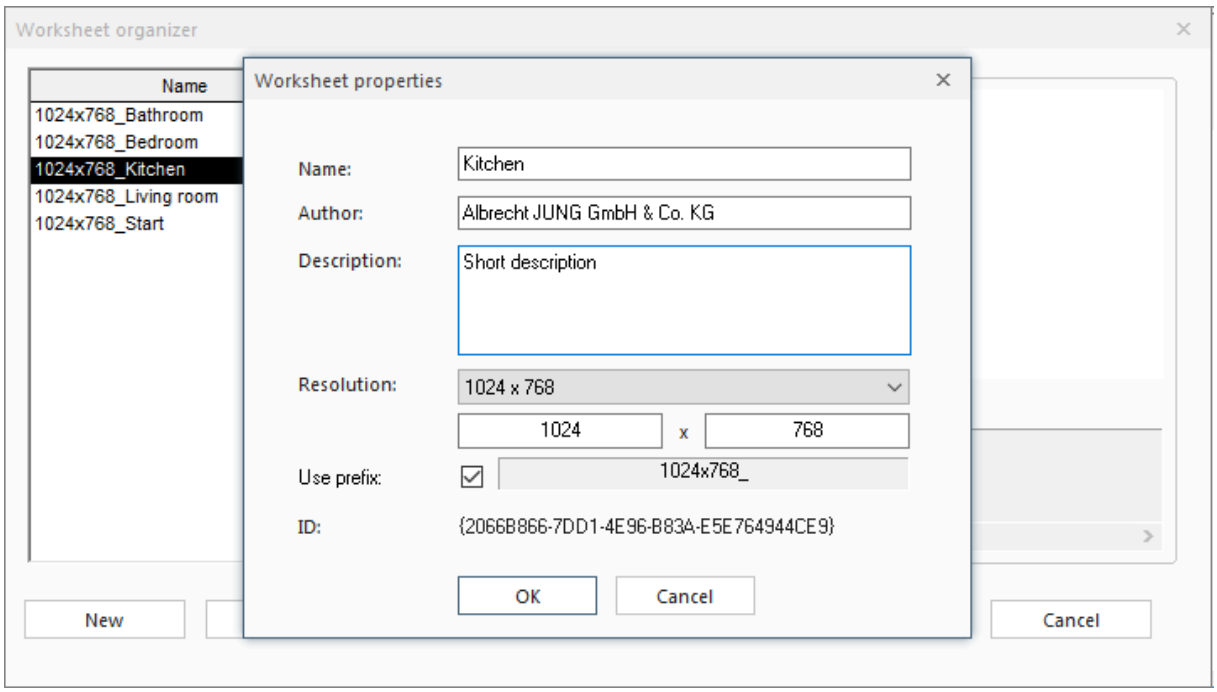
4.4 Worksheet Properties

In order to edit the properties of a worksheet, first open the worksheet in the visualization editor. Then open the Worksheet Organizer, select the opened worksheet and click the button ***Properties***.

The dialog ***Worksheet Properties*** opens where you can enter or edit the name of the worksheet, the author and the description. The name of the worksheet can only be edited when the according worksheet is closed.

4.4.1 Define Resolution

Here, you can set the resolution for individual worksheets with a name prefix. The prefix is necessary for the use of the worksheets by the visualization server. The resolution can only be changed if the corresponding worksheet is closed.



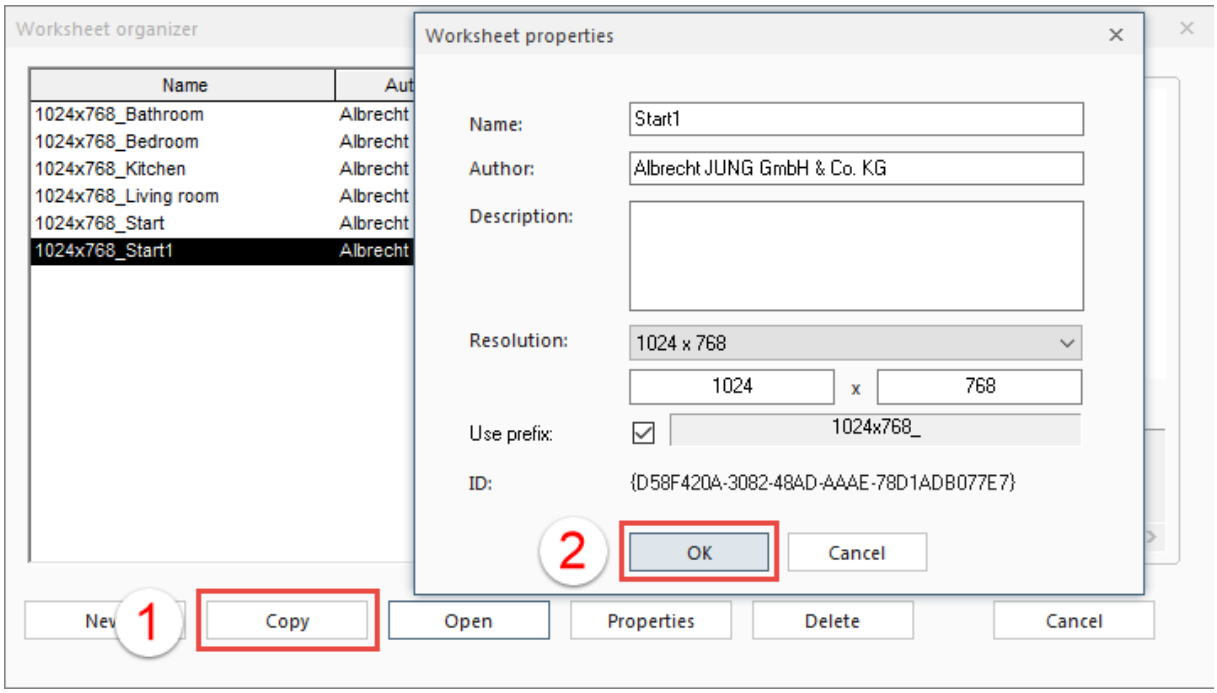
Click ***OK***, to close the dialog and to submit the changes. Otherwise click ***Cancel***.

Click on ***Cancel*** to cancel the action and to close the dialog ***Worksheet organizer***.

4.5 Copy the Worksheet

To copy a worksheet, you must first open the dialog Worksheet organizer. Click therefor under ***Worksheet*** on ***Worksheet Organizer***. Alternatively, choose the tool window ***Tasks*** and click under Worksheets on ***Organize***.

Select the desired worksheet and click on ***Copy***. The selected worksheet is copied and the dialog ***Worksheet properties*** opens.



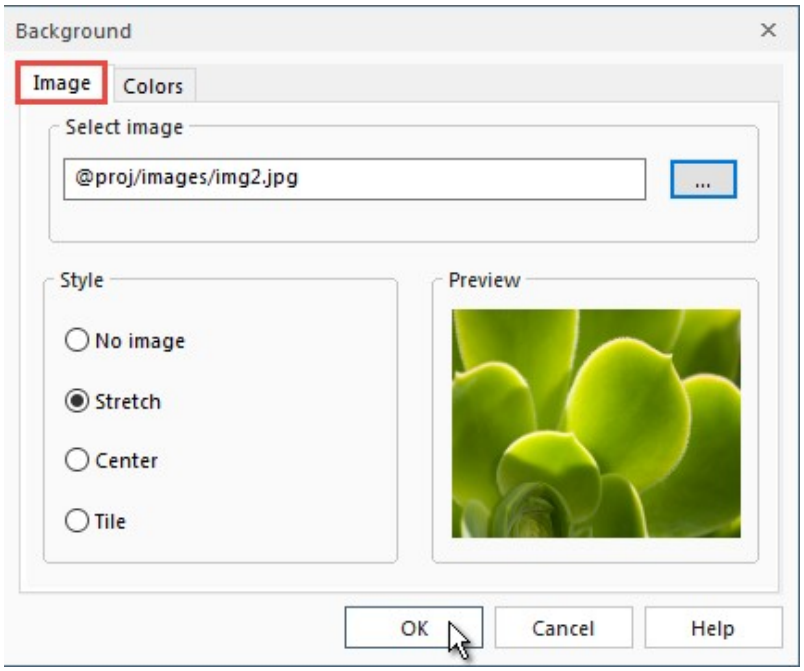
Here you can edit the name of the worksheet, the resolution, the author and the description. Else a consecutive numbering will be attached to the name of the worksheet.

4.6 Edit the Worksheet

4.6.1 Define Background

Choose the menu item ***Format*** and click ***Background***. The dialog ***Background*** opens.

Image:



To select a graphic for the active worksheet, choose the tab ***Image*** and click the browse button. The dialog ***Open*** opens. Select the desired image and click ***Open***. Supported formats are BMP, JPG, WMF and EMF.

Choose a ***Style*** and click on ***OK***.

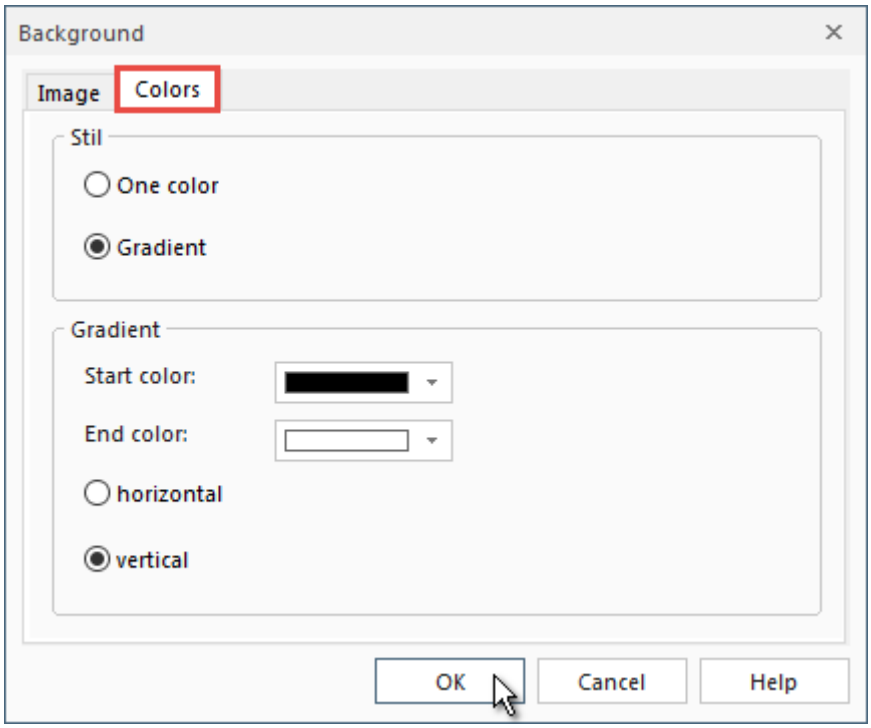
Note: *For to ensure that all graphics of the project are integrated during export in the control panel, all paths are converted into symbol paths. That way the path specifications do not have to be reconfigured if the project has been imported to a different computer.*

@sys: *This prefix replaces the complete path to the design directory under '**<Installation directory>/App/VisSysResources/Design/**'.*
Graphics with a path from this directory structure remain at this location.

@proj: *This prefix replaces the complete path to the project directory under '**<Installation directory>/Stations/local/projects/Project name .VIS/images/**'*
Graphics with a path from this directory structure are already in the project directory*.*

*All other graphics to be displayed in the visualization are automatically copied into the image directory of the current project **<@ proj / images>**.*

Colors:

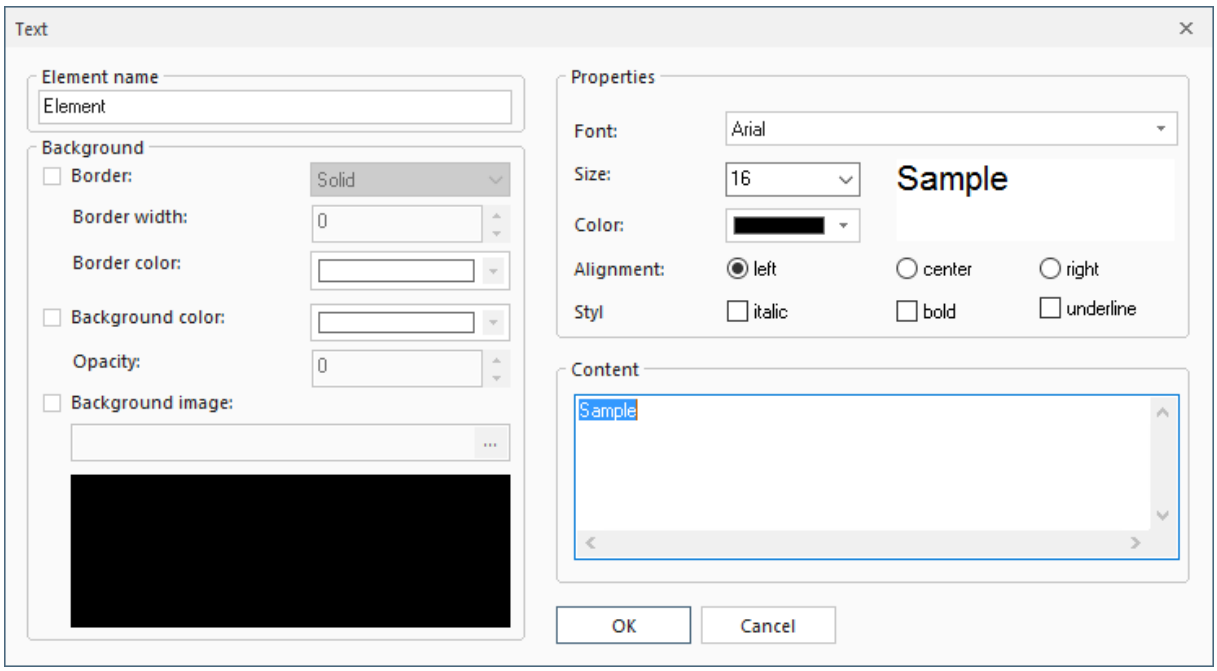


You can fill the background with a single color or a color gradient of two colors, either horizontally or vertically.

Click ***OK*** to submit the changes and to close the dialog.

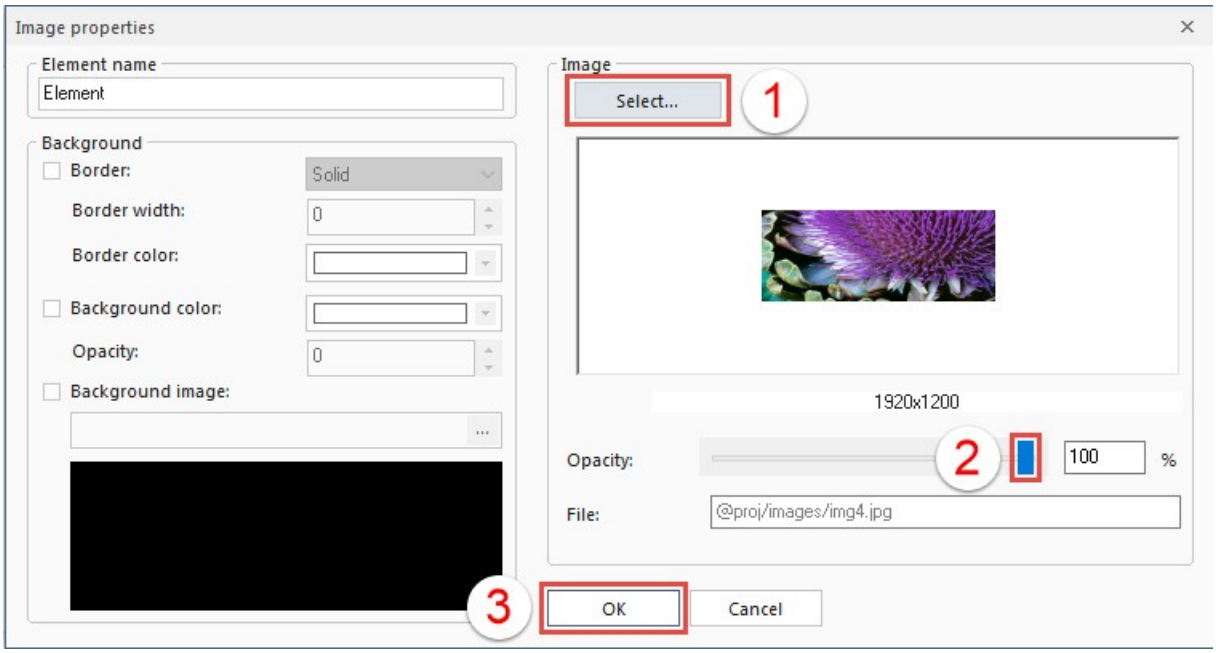
4.6.2 Insert Text Item

Click in the toolbar the button ***Text***. Then select in the worksheet a rectangular area in which the text item should be placed. The dialog ***Text*** opens where you can enter text into the contents field and define text styles. Click ***OK***.



4.6.3 Insert Image Item

Click in the toolbar the button ***Image*** and select in the worksheet the area in which the image will be displayed. The dialog ***Image properties*** opens. Click the button ***Browse*** to open the dialog ***Open*** where you can select an image. Select the desired image and click ***Open***.



Opacity: The slider is used to define the opacity of the image.

Supported file formats:

- Windows oder OS/2 Bitmap (*.bmp)
- CompuServe Graphics Interchange (*.gif)
- JPEG (*.jpg)
- Portable Network Graphics (*.png)
- Tagged Image File Format (*.tif)
- Windows Meta File (*.wmf)
- Windows Enhanced Meta File (*.emf)

Tips:

- *GIF and PNG images may be transparent (irrespective of the settings made in the dialog above), for example a light bulb with transparent pixels outside of the bulb's outline. In this case, the transparency will be retained in the worksheet.*
- *Example of use: You can insert a plain-colored bitmap which is semi-transparent on a background in order to highlight an area in the worksheet.*

Click ***OK*** to submit the settings and to close the dialog.

4.6.4 Animated Images

4.6.4.1 Image Library

A small selection of static and animated pictures will be installed by the setup.

> ***Note:*** *For to ensure that all graphics of the project are integrated during export in the control panel, all paths are converted into symbol paths. That way the path specifications do not have to be reconfigured if the project has been imported to a different computer.*
>
> ***@sys:*** *This prefix replaces the complete path to the design directory under '**<Installation directory>/App/VisSysResources/Design/**'.*
> *Graphics with a path from this directory structure remain at this location.*
>
> ***@proj:*** *This prefix replaces the complete path to the project directory under '**<Installation directory>/Stations/local/projects/Project name .VIS/images/**'*
> *Graphics with a path from this directory structure are* already in the project directory.
>
> *All other graphics to be displayed in the visualization are automatically copied into the image directory of the current project **<@ proj / images>**.*

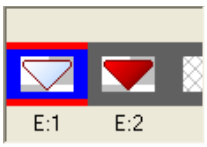
4.6.4.2 Example: Analog Image Output

Similarly to the analog image input, the ***analog image output*** is able to use animated GIFs to display switching operations.

Additionally to the animated GIF, the user can assign values which will be sent to the process by clicking with the left mouse button on the parameter display. It is possible to specify a value to send on short click and long click.

Depending on the value settings, it is possible to create special displays like a KNX dimmer switch or a KNX blind control

The following example shows the usage of three different analog image outputs (**arrow up**, **arrow down** and **crossbar**) together with an analog image input (window with blind) to control and display blinds.

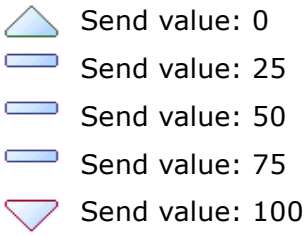


The figure on the left side shows the pictures which are stored in one of the animated GIFs (arrow down).

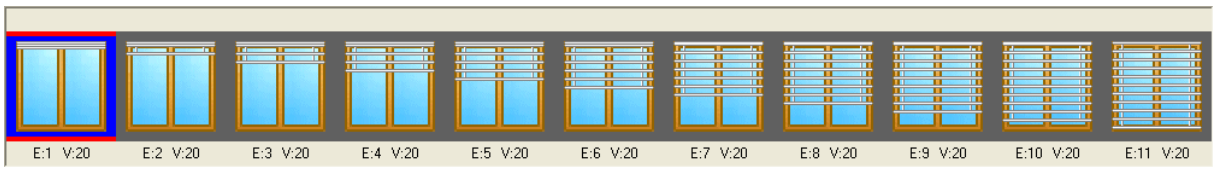
The figure on the right side shows the three animations: arrow up, crossbar and arrow down. If a long mouse click is detected (left mouse button clicked and held down), the animations will be played and the configured analog value will be sent to the process (0% for arrow up, and 100% for arrow down in our sample below).

Example of use:

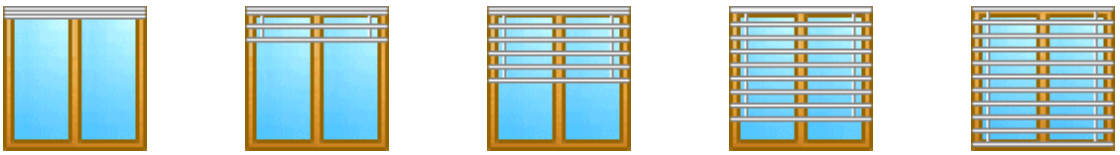
Choose the tool window ***Catalog*** and under the category ***Parameter displays*** the folder ***Analog outputs***. Then click the parameter display ***Analog output*** with the left mouse button and drag it into the worksheet. The property dialog of the display opens. Select the images and then enter the values to be sent to the process (the pictures are available in the directory *Bitmaps*: BtnUpGreen.gif, BtnBlue.gif, BtnDownRed.gif).



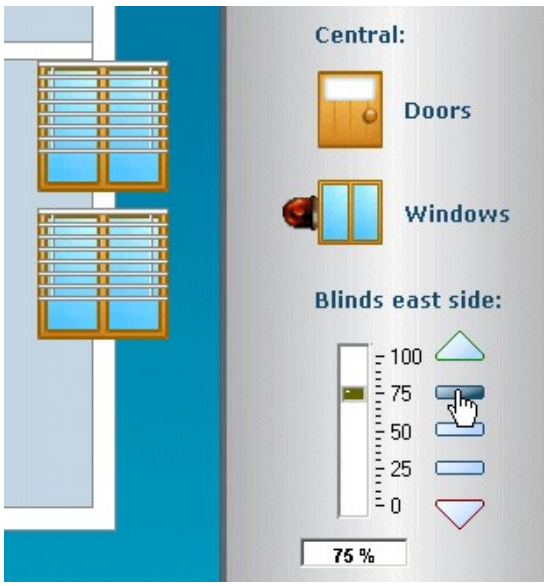
Additionally, you can create an analog image input with the picture 'Anim_Blind.gif' (available in the directory *Bitmaps*).



Result:



Value 0 Value 25 Value 50 Value 75 Value 100

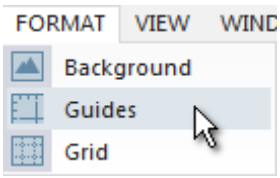


The figure on the left shows a part of a worksheet which contains the sample above.

4.6.5 Using Grid and Guides

Click in the toolbar the button ***Guides*** to toggle the visibility of guide lines.

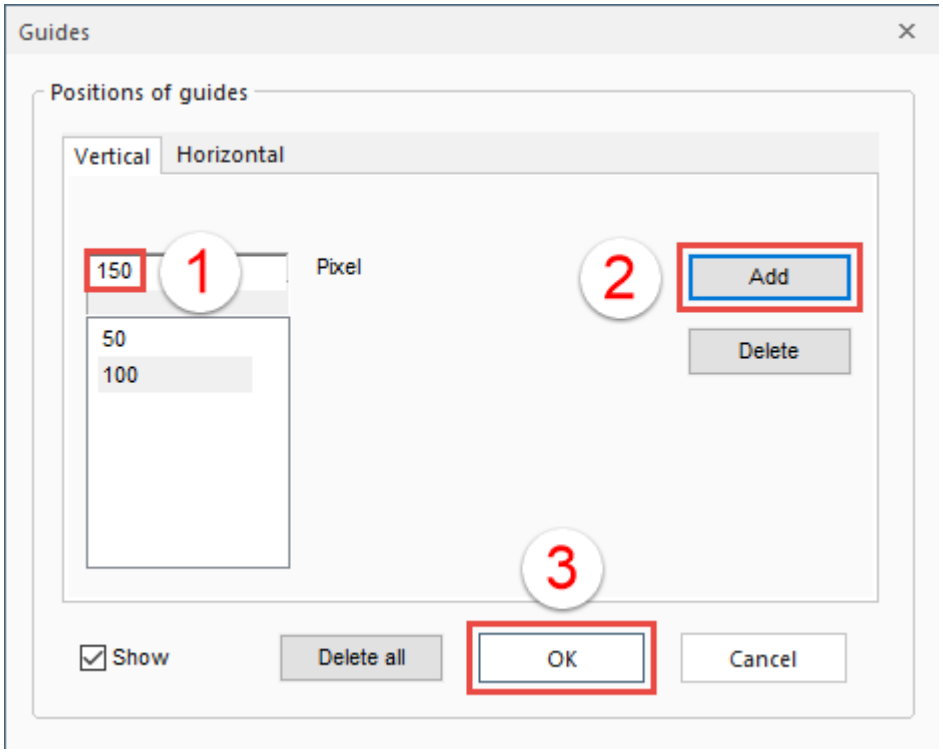
Click in the toolbar the button ***Grid*** to toggle the visibility of the grid.



Under the menu item ***Format***, ***Guides*** and ***Grid*** *can be configured.*

Guides:

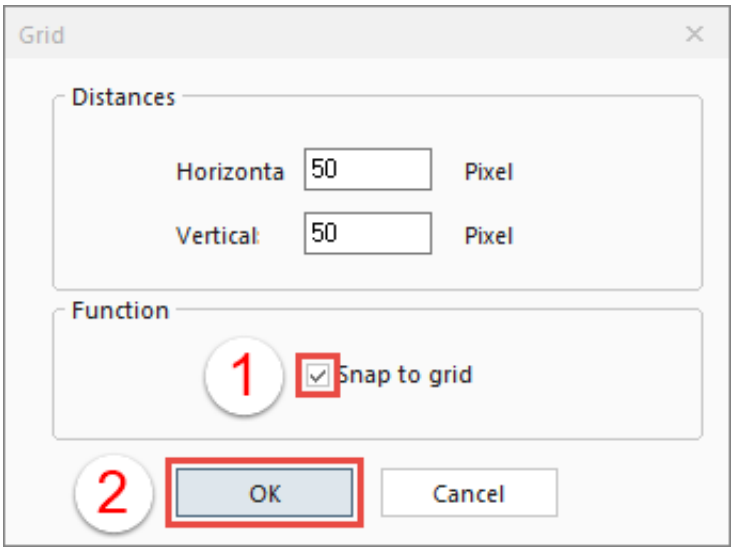
Enter the horizontal and vertical positions of the guides and click the button ***Add***. The new guide is displayed at the defined position.



To delete a guide, select the guide and click the button ***Delete***. To delete all guides, click ***Delete all***.

Grid:

In the dialog ***Grid*** the horizontal and vertical grid distances can be defined independently.



The **snap-to-grid function** enables selected or inserted items to snap automatically with the top-left corner of the item to the closest grid point. To enable this function, select the checkbox ***Snap to grid***.

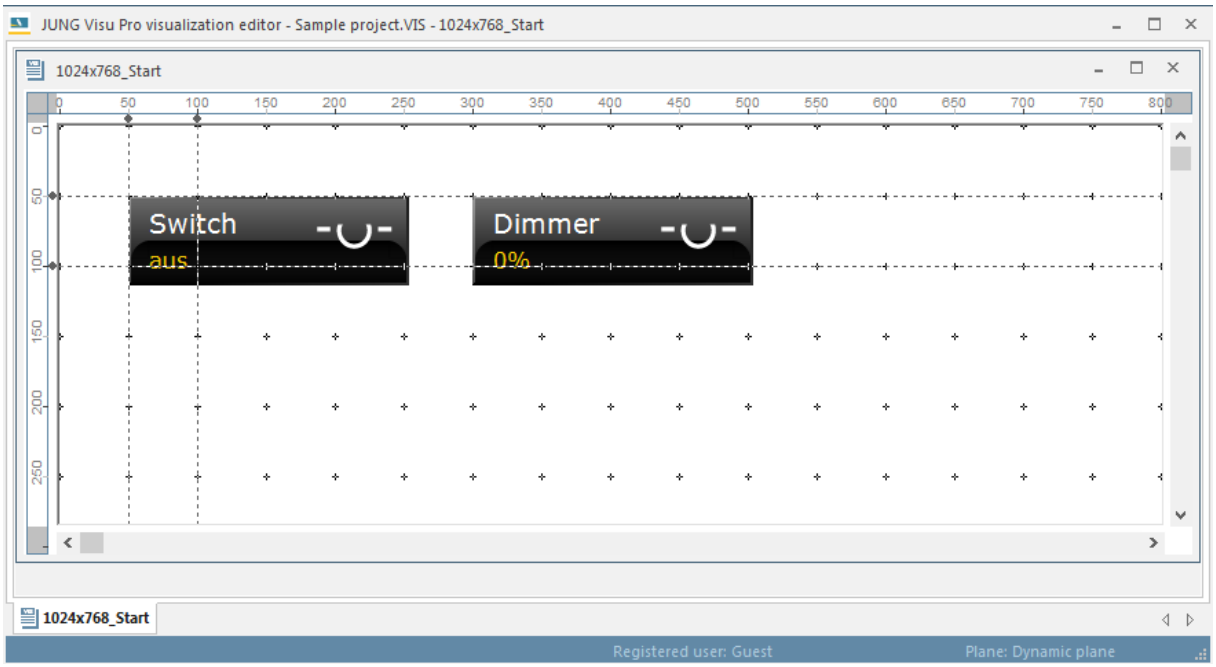
Click ***OK*** to submit the settings and to close the dialog.

4.6.6 Moving Elements with Keyboard Keys

You can select elements by clicking on them. For a multiple selection hold down the Ctrl key or draw a selection rectangle over the elements you want to select. Move them with the keyboard as shown in the following.

Key combinations:

← → ↑ ↓	Move one pixel in any direction.
⇧ + ← → ↑ ↓	Move one grid point in any direction.
Ctrl + ← →	Align left margin with next vertical guide.
Ctrl + ↑ ↓	Align top margin with next horizontal guide.
Ctrl + ⇧ + ← →	Align right margin with next vertical guide.
Ctrl + ⇧ + ↑ ↓	Align bottom margin with next horizontal guide.



4.7 Checking Functions

General checking options:

Check the connection of the visualization project to the logical process model:

- Is a logical process model connected to the visualization project?
- Is the connected logical process model still available?

Check the connections of the display elements to the process variables:

- Are the display elements properly connected to process variables?
- Are the connected process variables still available?
- Have the properties of the process variables changed in such a way that they do not match the parameter displays anymore?

Check the dimensions of the display elements:

- Were display items placed outside of the worksheet bounds?
- Is a display element too small and possibly not visible?

Check the names of the worksheets:

- Are the specified resolution prefixes valid?

Check the connections to the buttons:

Action: Switch tab panel

- Are the specified tab panels available?
- Are the specified cell names available?

Action: View worksheet

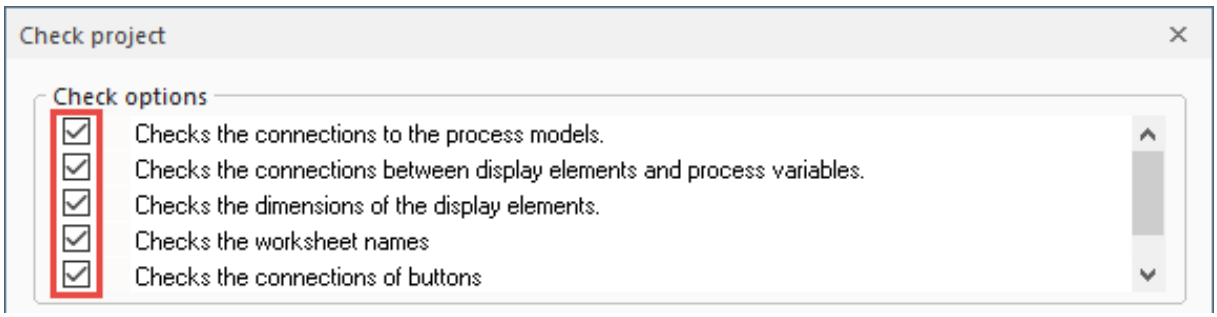
- Are the specified worksheets available?

These checks are optional and can be selected independently. As a result of the check procedure a report with warnings and errors is displayed in the message area at the bottom of the worksheet editor.

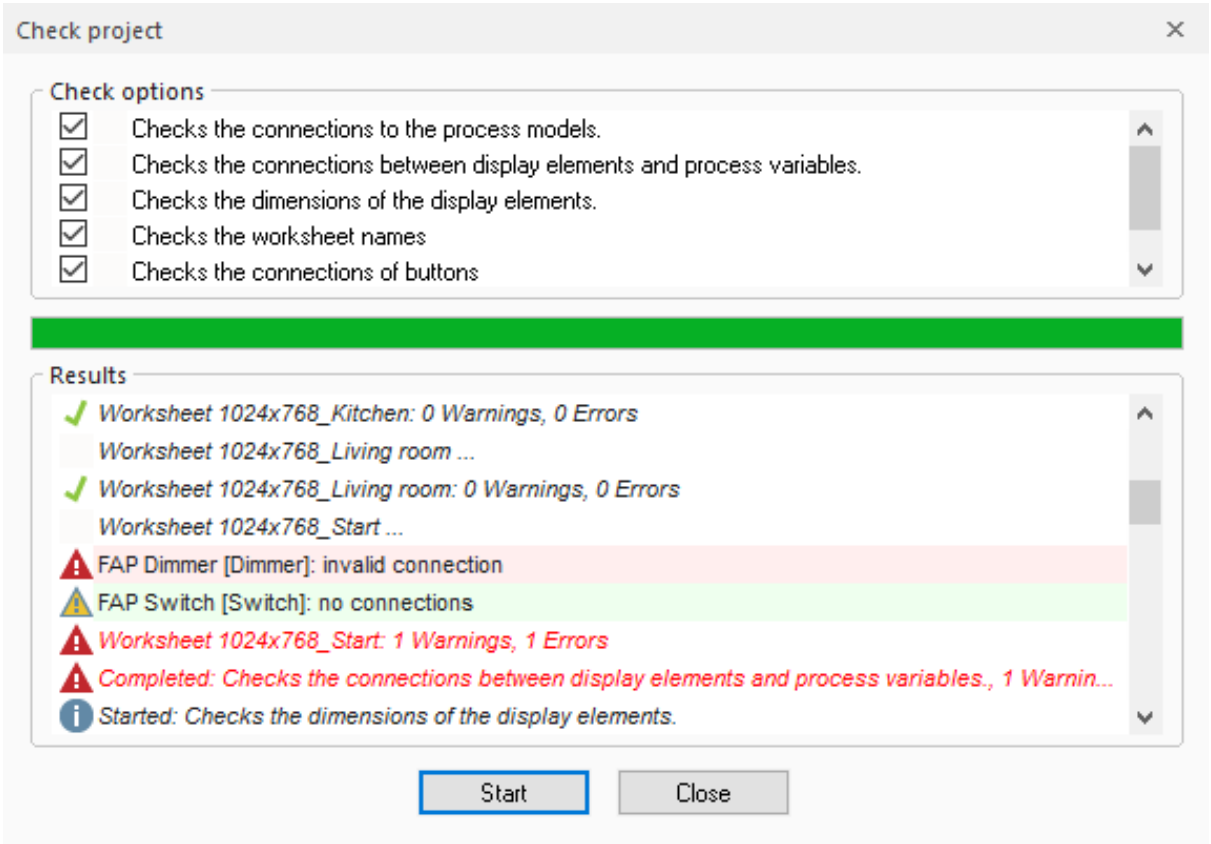
For errors of single items, a double click on the error or warning message will select the item. If necessary, the respective worksheet will be opened.

4.7.1 Check Project

Open the visualization project you wish to check. Then choose the menu item ***File*** and click ***Check***. Alternatively, click the button ***Check*** in the toolbar. The dialog ***Check project*** opens.

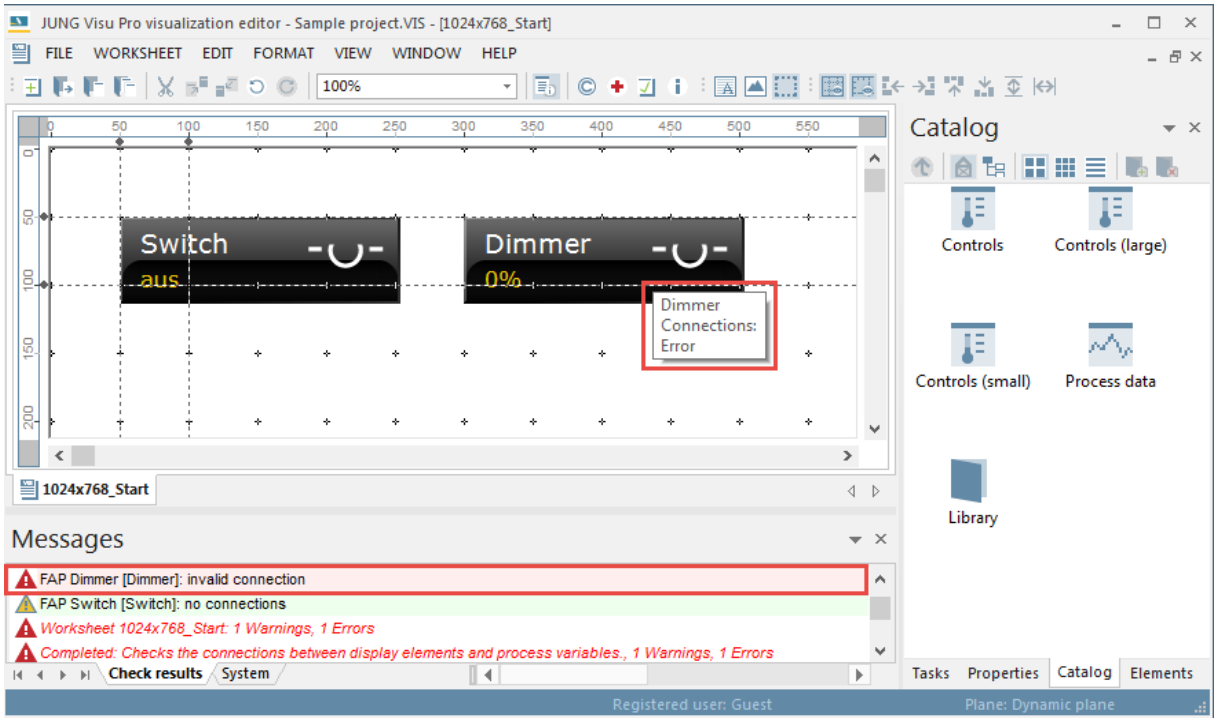


Select the desired options for the project check and press the button ***Start***.

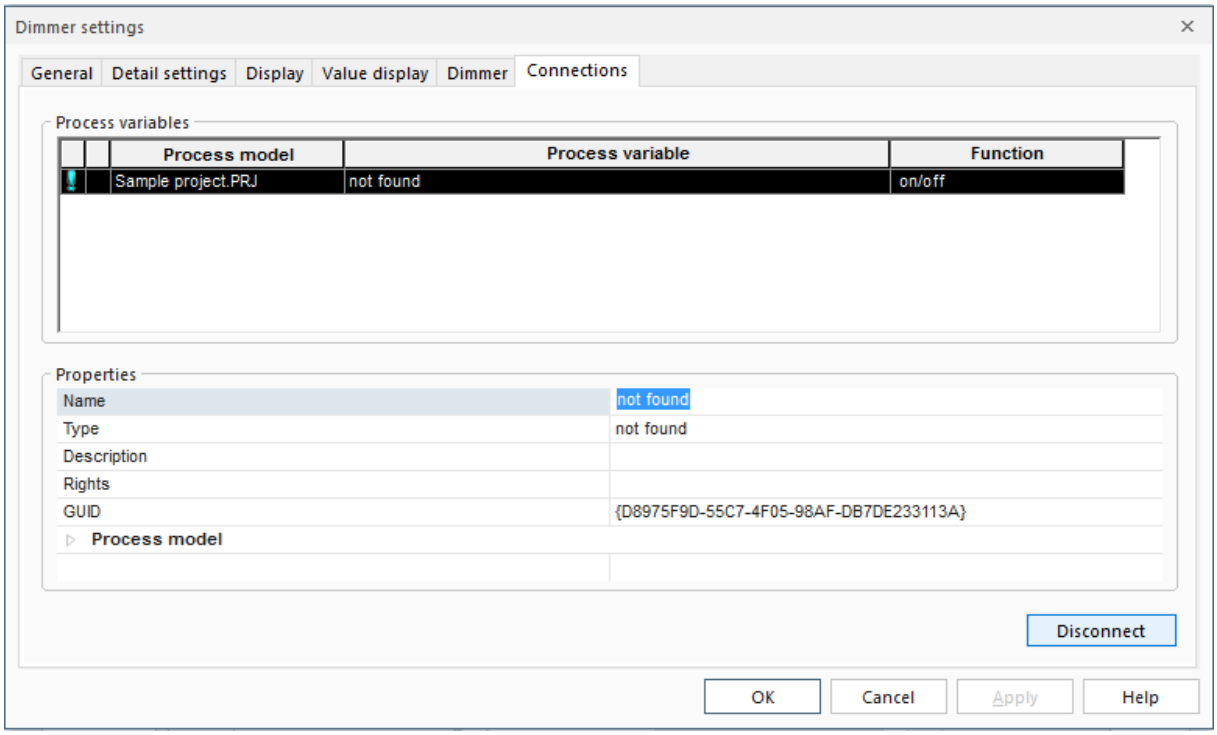


After the check has been completed, click the button ***Cancel*** to close the dialog.

To work on the individual errors and warnings, first click the toolbar button to open the message window at the bottom of the visualization editor. If the error or warning relates to an individual display item, double-click the message to bring up the item.



Example: In the example above the connection of a display element to a process variable is not correct. The respective worksheet has been opened and the item has been selected by a double click on the message.
Now double-click the display element to open the property dialog of the element and correct the problem.

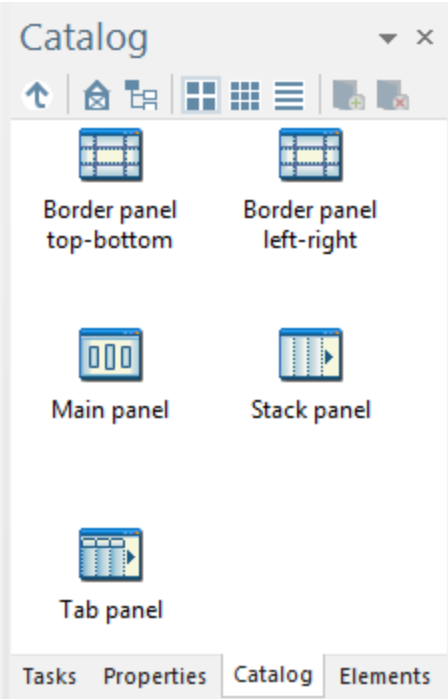


In the property dialog of the display element choose the tab ***Connections***. It shows all process variables connected with the display element. To disconnect the problem causing process variable, first select the process variable and then click the button ***Disconnect***.

5 Elements

5.1 Panels

The panels are located in the tool window ***Catalog*** in the directory ***Controls*** in the sub directory ***Panels*** from where they can be inserted into the worksheet.



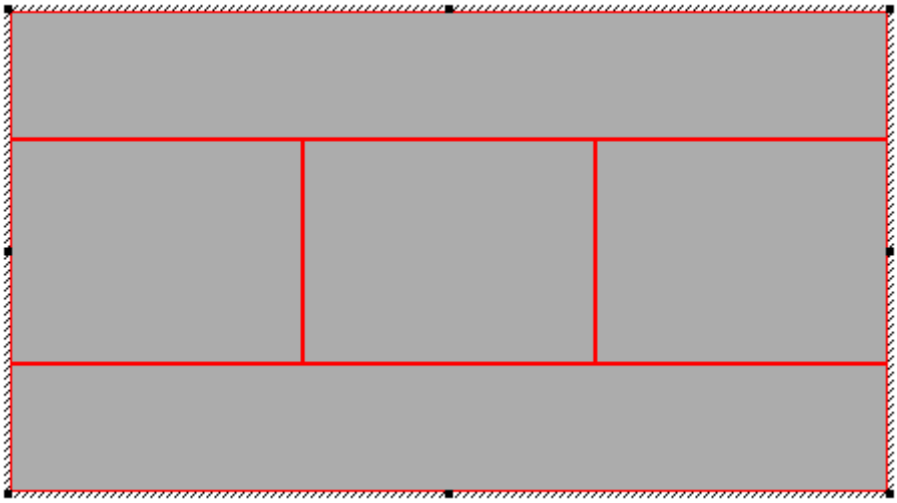
Panels are visual containers which group elements. Every panel contains cells in which display elements or other panels can be inserted by means of the Drag and Drop function.

To insert a panel into a worksheet, press and hold down the left mouse button on the panel and drag it into the worksheet.

Once a panel has been inserted into the worksheet, the property dialog opens where you can define the settings for the panel.

You can scale the panel up or down to facilitate its configuration. Elements can be inserted into the cells which are marked by red frames.

To edit a panel, select it first. A selected panel is enclosed by a white rectangle with black dots at each corner and side.

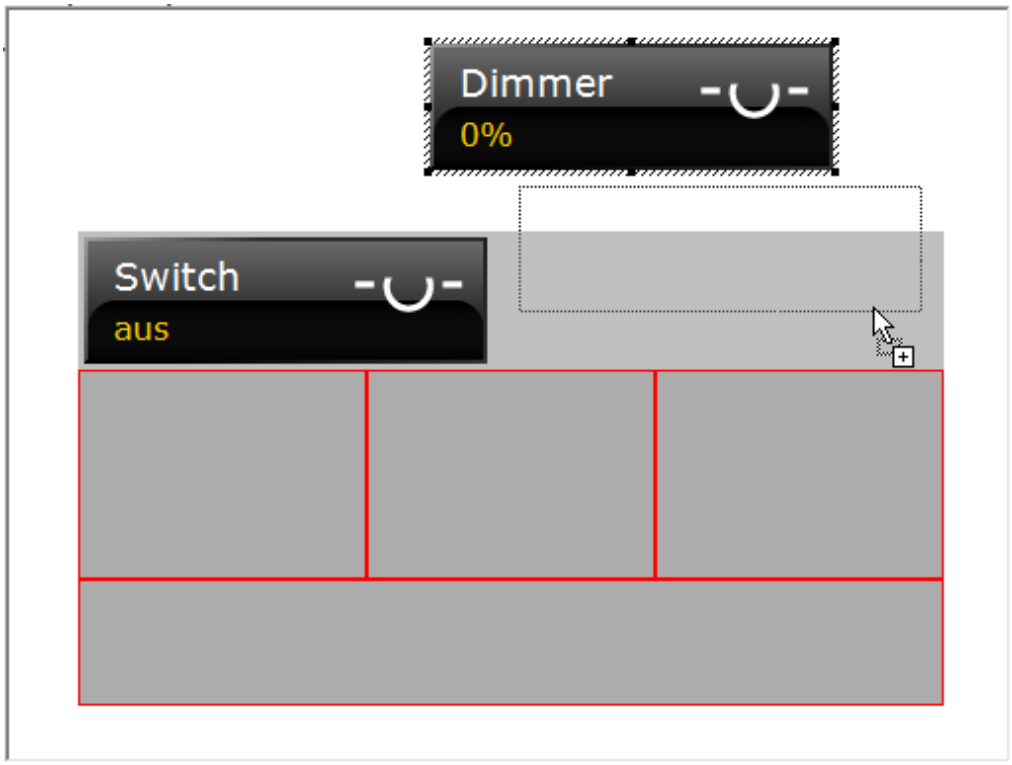


You can drag the black dots to resize the panel.

Insert an element into a cell

Each cell can host one object (display element or panel). To insert an object into a cell, drag the object first to a free area in the worksheet. The property dialog of the dragged object opens automatically.

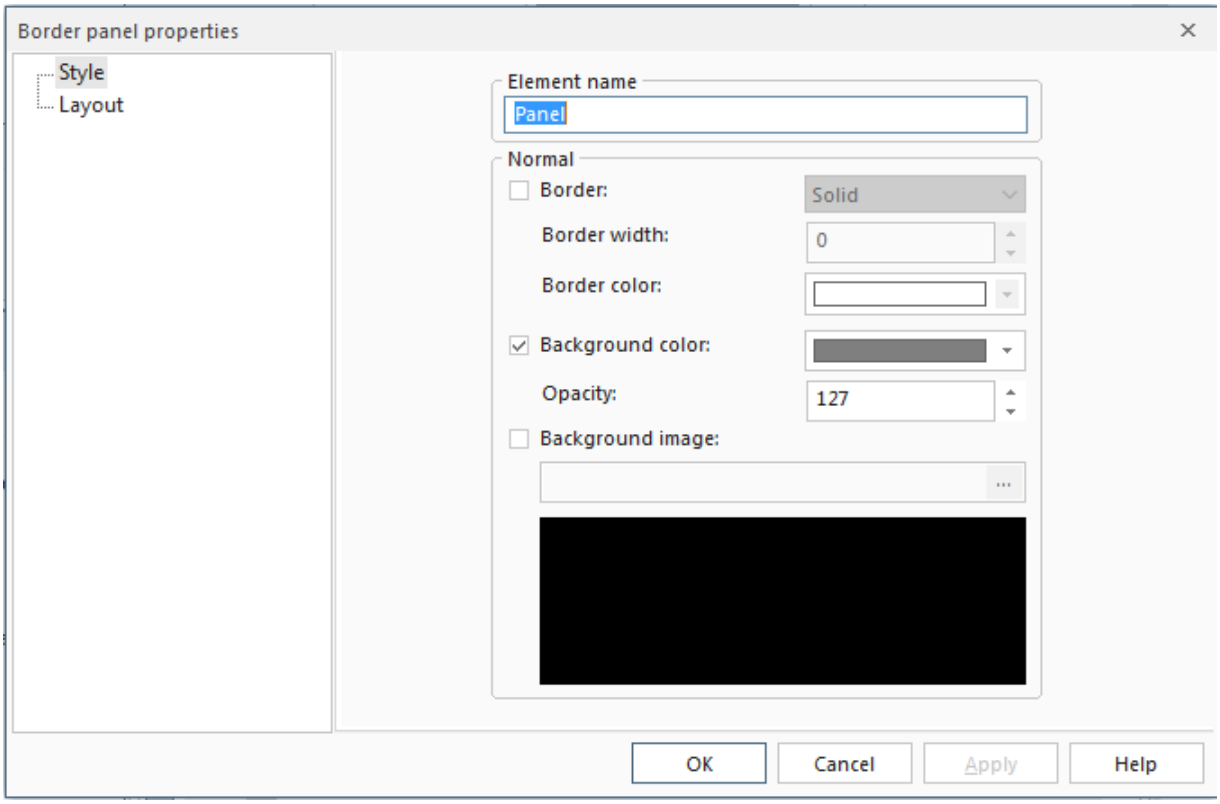
Define the desired settings for the object. Then drag the object into the desired cell and drop it.



The element docks at the cell on which the mouse cursor is currently over.

5.1.1 Style Options

For the layout of the panels, there are various design options. These can be found in the properties under ***Style***.



Element name: The element name is used for the internal process when editing projects in the Visualization Editor, and is not considered in the presentation.

Border: Appearance, width and color of the outer frame of a panel can be set here.

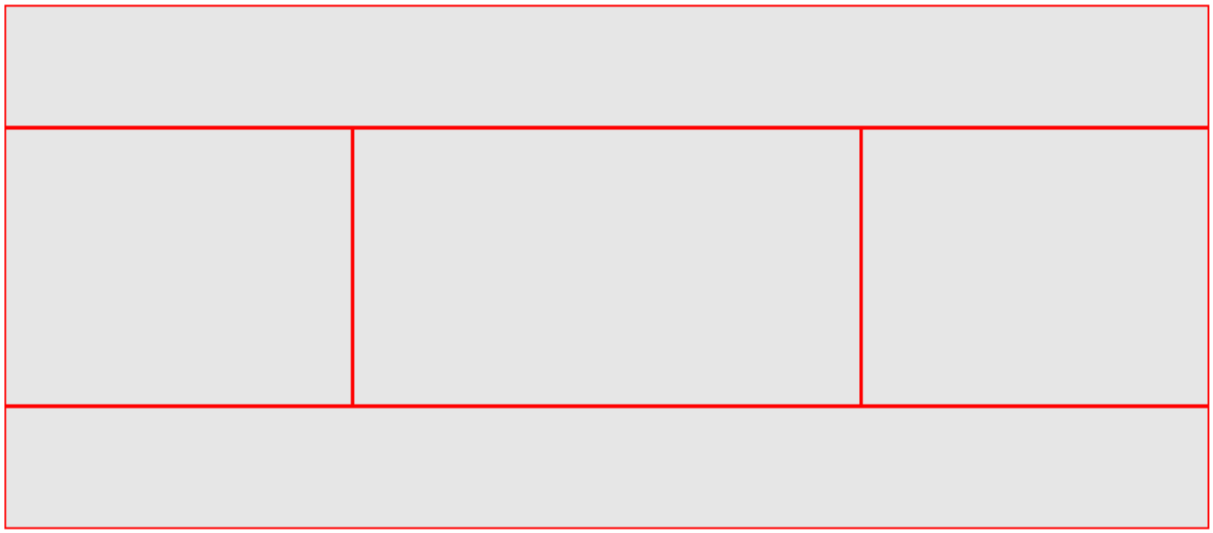
Background color: The background color is covered when a background image is set.

Background image: The background image is automatically scaled according to the size of the panel. It fills the entire background of the panel.

5.1.2 Border Panel

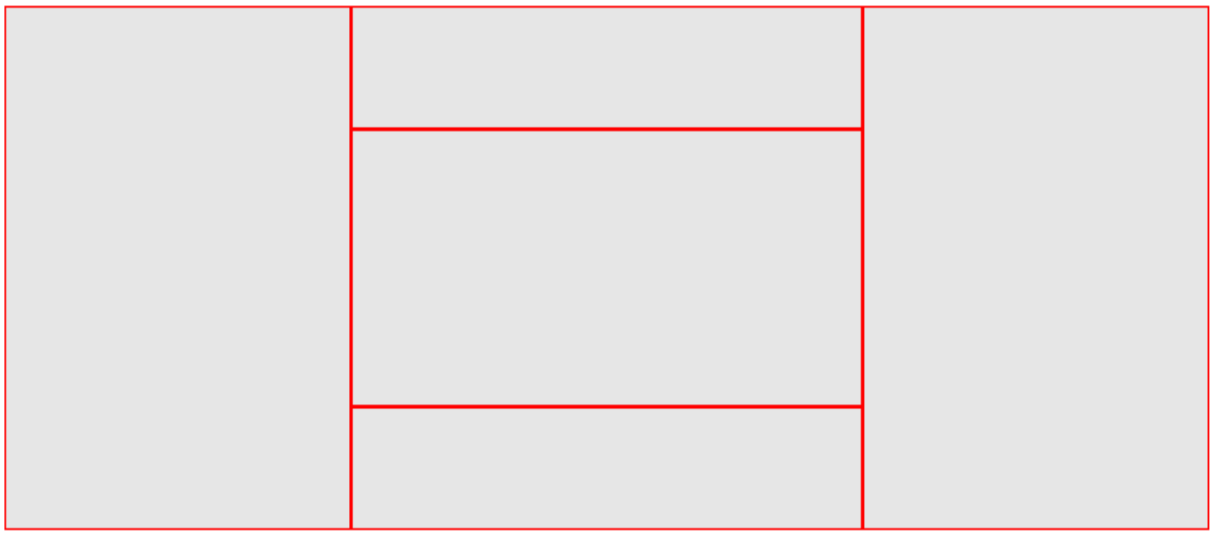
The border panel contains by default four cells: at the top, bottom, right and left side. There are two versions of the border panel:

Border panel top-bottom



In the ***border panel top-bottom***, the cells at the top and bottom side of the panel extend over the entire width of the panel.

Border panel left-right



In the ***border panel left-right,*** the left and right cells extend over the entire height of the panel.

LAYOUT PROPERTIES

Visible cells

Define which cells are to be shown in the panel. To hide a cell, clear the corresponding checkbox.

> **Note**
>
> **Preferred size**: If you insert into the cells of the border panel elements with different preferred sizes, the preferred size is allocated to the elements as long as the border panel offers enough space. If the border panel does not provide enough space, the preferred size of elements is ignored and the available space is distributed equally among all elements.

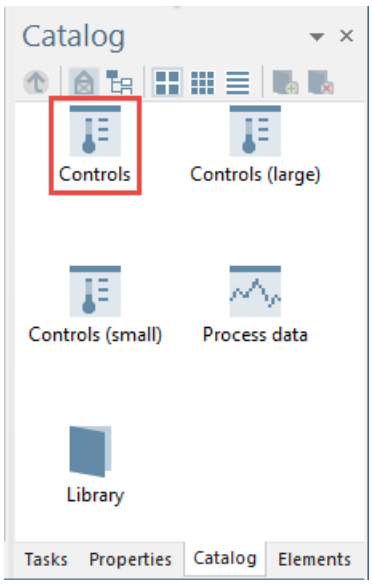
5.1.3 Main Panel

The main panel contains three cells, the size of which can be configured. The three cells of the main panel can be rotated (toggled) in the visualization server. For this purpose, click in the server on any point to the right of the center cell to initiate the clockwise rotation. To initiate the counter clockwise rotation, click on any point to the left of the center cell.

LAYOUT PROPERTIES

Cell size: Indicate the size of the cells in unit. Default: 1 unit = 64 pixels.

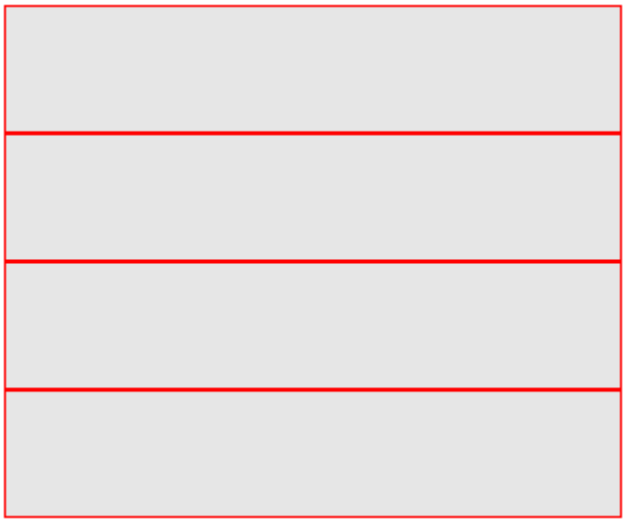
Note



You find the controls in the tool window ***Catalog*** in the directory ***Controls***.

5.1.4 Stack Panel

In the stack panel the cells can be stacked vertically or horizontally. The number of the cells is configurable.



LAYOUT PROPERTIES

Alignment: The cells in the stack panel can be stacked vertically or horizontally.

Decoration: Select the option ***Show scroll arrows*** to show scroll arrows in the server.

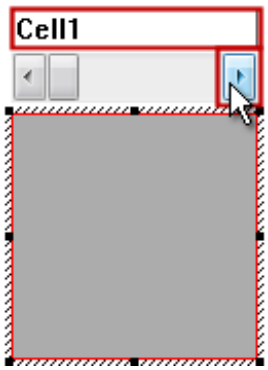
Cells: Set the number of cells. *Default: 4*.

5.1.4.1 SCIP Stack Panel

The SCIP stack panel is a copy of the common stack panel except for the difference that it is adapted for Smart Control IP devices.

5.1.5 Tab Panel

A tab panel contains cells which function as tabs in the visualization server. To be able to toggle the tabs of a tab panel, the control ***Button*** must be inserted in the worksheet. You can assign different actions to the control *Button* which will be initiated in the visualization server when the button is clicked. To initiate the toggle function, assign the function ***Toggle tab panel*** to the button.



You can configure the number of the switchable tabs in the property dialog of the tab panel.

Configure a tab panel

When the tab panel is selected in the visualization editor, an input field and a scrollbar are shown above the panel. The input field serves to enter the names of the tabs. With the scrollbar you can switch from one tab to the next.

CELLS

Set the number of cells here. Actualize the list of cells with the button ***Apply***. To rename a cell, select it and press the ***F2 key*** to enter a new name.
The cells work like tabs which can be switched in the visualization server.

A cell name may occur only once per worksheet.

Rules for the assignment of cell names:

- The following characters are allowed:
  [A-a] to [Z-z]
  [0-9]
  [_] underscore
- The following words are **not** allowed:
  and, end, in, repeat, while, break, false, local, return, do, for, nil, then, else, function, not, true, elseif, if, or, until
- The cell name **must not** start with a digit
- Element names are **case sensitive**

5.2 Controls

5.2.1 General Settings

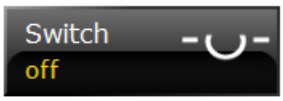
You can choose between three sizes of control elements:

Elements:

normal display

Elements large:

larger display

Elements small: 

smaller display

The following section provides information on general settings which can be configured in the property dialog of every control. The options vary by control element type.

5.2.1.1 Tab *General*

5.2.1.1.1 Element Name

The element name is an important specification which facilitates the editing of projects. It serves the internal processing of visualization projects and is not displayed in the visualization server.

The tab ***Elements*** in the tool window ***Elements*** shows all inserted controls under their element name. It is important to assign appropriate element names to the display elements in order to have a better overview over all display elements contained in a worksheet and to facilitate the editing of the worksheets.

An element name **must** be assigned!

You can edit the element name in the property dialog of the display element or in the tool window ***Elements*** on the tab ***Elements***.

An element name may occur only once per worksheet.

Rules for the assignment of element names:

- The following characters are allowed:

  [A-a] to [Z-z]
  [0-9]
  [_] underscore

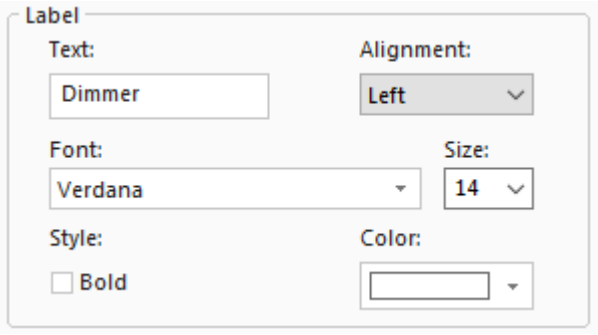
- The following words are **not** allowed:
  and, end, in, repeat, while, break, false, local, return, do, for, nil, then, else, function, not, true, elseif, if, or, until

- The element name **must not** start with a digit

- Element names are **case sensitive**

5.2.1.1.2 Label

Every display element can be labeled. The label is shown on the display element if the layout is configured accordingly.



Text: Enter the label.

Alignment: The label can be displayed left-aligned, right-aligned or centered. *Default: Left*.

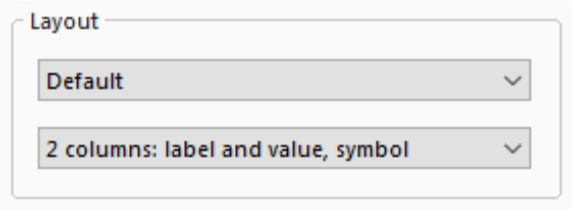
Font: Select the desired font and font size.

Style: Select the option ***Bold*** to display the label in bold style. *Default: Disabled*.

Color: Select a color from the dropdown list. *Default: White*.

5.2.1.1.3 Layout

Under ***Layout*** you can define the appearance of the control elements, if any is available for this control element.

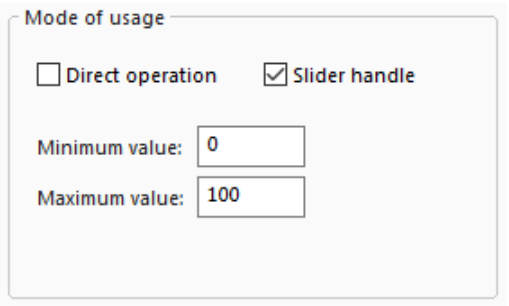


In the first drop-down menu you can set the display size for the control element.
In addition to the ***Default***, there is the special sizes ***Smart Control IP*** and ***Small***.

In the second drop-down menu you can specify whether and in which order the information label, value and symbol are shown in the display element.

5.2.1.1.4 Mode of Usage

The ***Mode of usage*** determines how the display element is operated in the visualization server. For analog display elements one of the following modes can be selected:



- **Direct operation**
- **Operation via slider**
- **Operation via slider handle**

Direct operation: The action is initiated when the display element is clicked with the left mouse button.

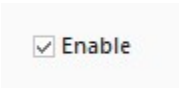
Operation via slider: To control a display element via slider, **clear** the checkbox ***Direct control***. Then the slider will be activated when the display element is selected in the server. To increase or decrease the current value, the upper or lower part of the slider is clicked.

Operation via slider handle: When the display element is selected in the server, the slider with the slider handle is activated. The current value can be increased or decreased by moving the slider handle up or down.

Minimum/Maximum value: Enter the upper and lower limit of the value range.

5.2.1.2 Tab *Detail Settings*

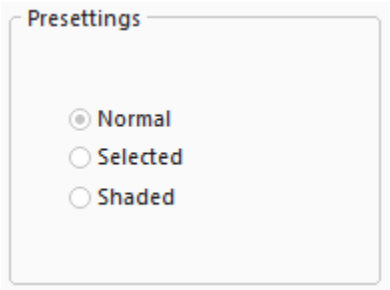
5.2.1.2.1 Enable



Here you can enable or disable the display element. *Default: Enabled*. If the element is disabled, it does not initiate any action when clicked.

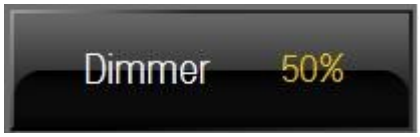
A display element can be configured in the disabled state. But in the visualization server it will always be displayed in the undefined state and pressing it will have no effect.

5.2.1.2.2 Presetting



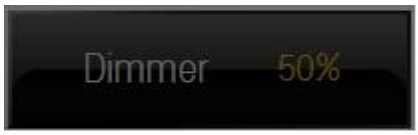
Under ***Presetting*** you can define how the display element appears after the worksheet has been loaded.

Select one of the following options:



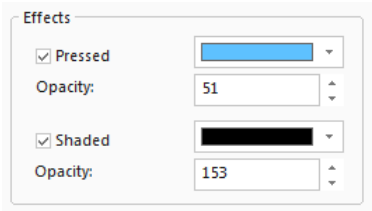
Normal: The display element has default appearance.

Selected: The display element is displayed brighter and is enclosed by a blue frame.



Shaded: The display element is shaded.

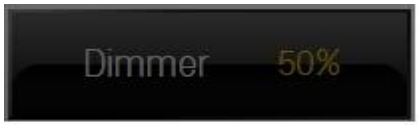
5.2.1.2.3 Effects



Here you can define whether the display element provides a visual feedback when clicked. The effects merely serve to visualize mouse events and do not have any impact on the sending of commands.

The following effects can be selected:

Pressed: When pressed for a longer period of time, a semi-transparent blue rectangle is laid over the display element.



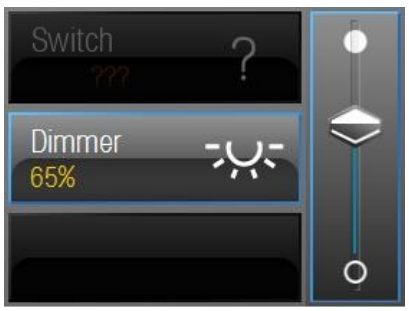
Shaded: The display element is shaded when it is not selected within its selection group.

5.2.1.2.4 Selection Group

Display elements can be arranged in selection groups. A selection group enables the user to select one display element out of a number of display elements in order to initiate a certain action. The behavior of a display element within its selection group depends on the behavior of the other elements in this group: if you select one display element and thereby activate it, the display elements which are not selected, are deactivated.

The selected element is enclosed by a light blue frame. The elements that are not selected, are displayed normally or are shaded (provided the effect ***Shaded*** is enabled).

Within a selection group only one display element may be selected.

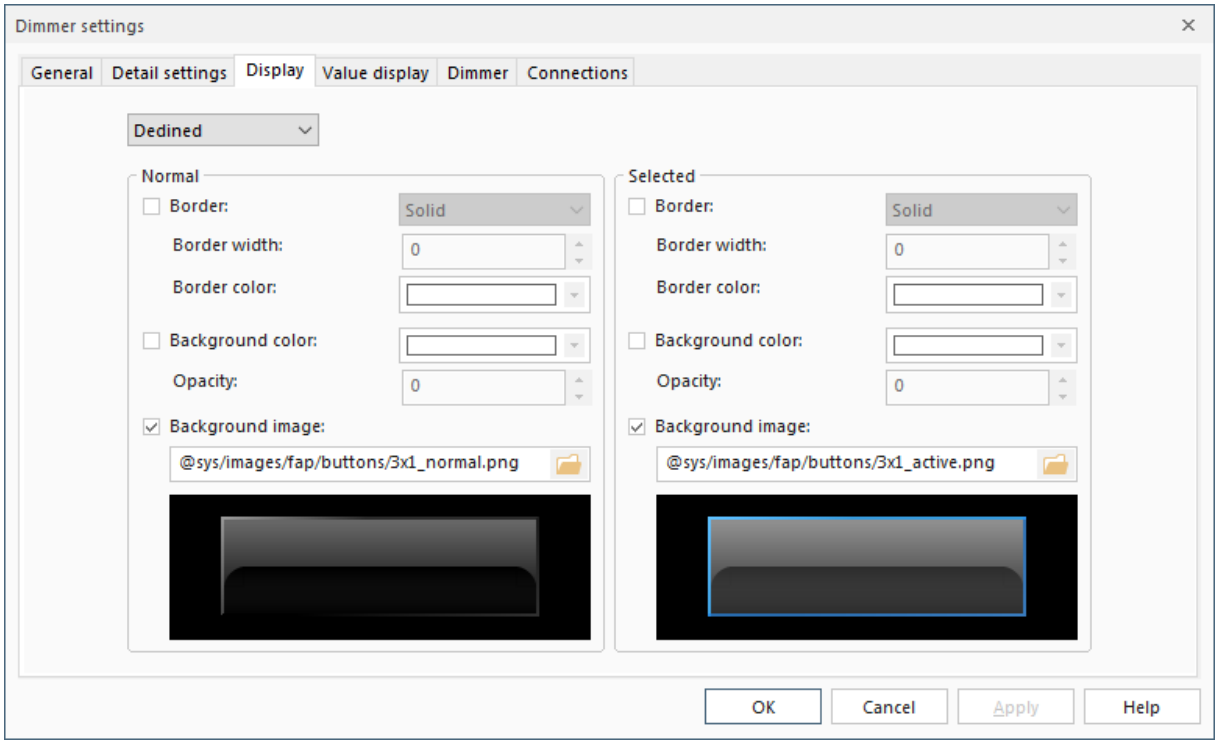


In this example, three display elements are assigned to the same selection group. The display element with the label ***Dimmer*** is selected. Consequently, the other two elements are shaded (provided that the effect ***Shaded*** is enabled).

The slider is activated for the selected element. In this example, the slider refers to the display element *Dimmer*.

To assign a display element to a selection group, select the checkbox ***Selection group***. Then enter in the input field ***Group name*** the selection group or select an already existing group from the combo box.

5.2.1.3 Tab *Display*



In the Tab Display can be specified how the border and background of an element are displayed in various states.

Border: Choose here the border settings for the particular status.

Background color: Select here the background color for the particular status. If a background image has been set, it covers the background color.

Background image: Select here a background image for the particular status.

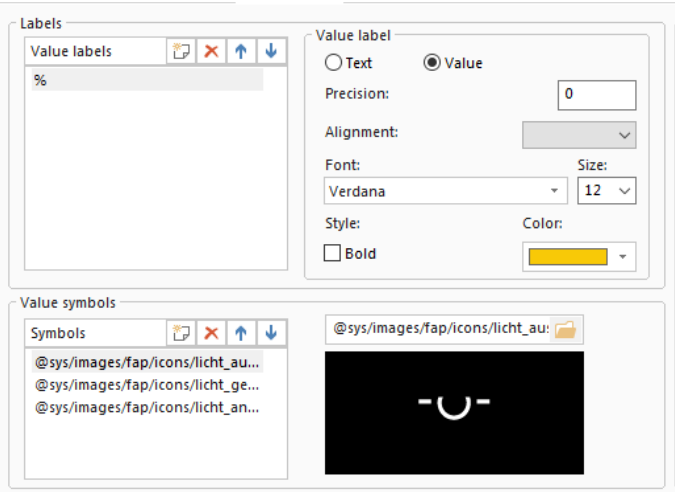
If **nothing** is activated, the element will be transparent.

5.2.1.4 Tab *Value Display*

On the tab ***Value display*** you can define how the values received from the bus are to be displayed on the display element.

Value labels

When creating value labels, you can define whether the display element shows the current value (***Value***) or a text (***Text***) in place of the value.

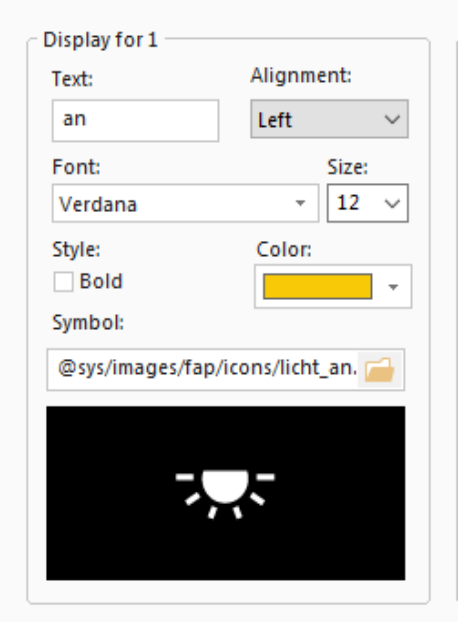


Text: Select the radio button ***Text***, if you want a text to be displayed instead of the value.

Value: Select the radio button ***Value***, if you want the display element to display the current value. A unit for the value is defined by default. You can edit the unit in the input field under ***Value labels***.

Note: The value labels are listed in ascending order: the lowest value is listed first and the highest value last.

Display for a *certain value*



Here you can configure the display for the value **1=on**.

Text: Enter the text for the value **1**. *Default: On*.

Alignment: The value label can be displayed left-aligned, right-aligned or centered. *Default: Left*.

Font: Select the desired font and font size.

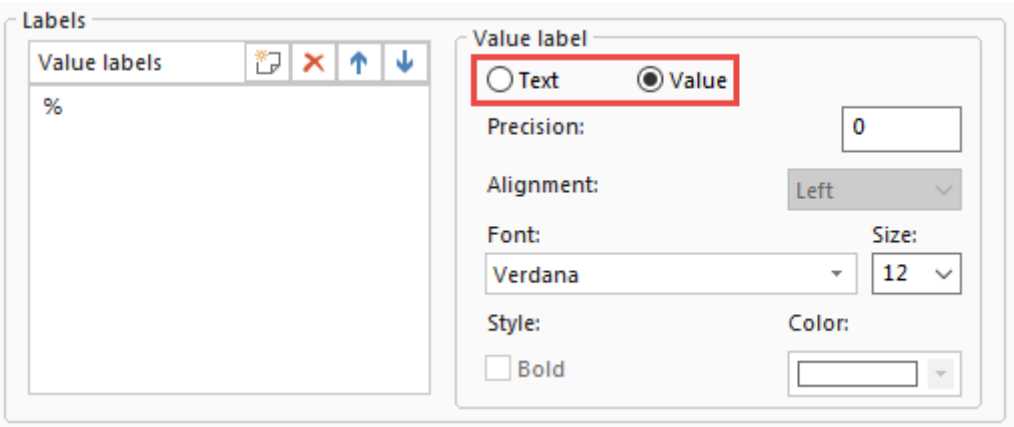
Style: The label font can be displayed in bold style. *Default: Disabled*.

Color: Select a font color from the dropdown list. *Default: Yellow*.

Symbol: Select a symbol for the value 1: Enter in the input field the name of the image file or click the browse button and select the desired image from the default directory. The selected symbol is displayed in the preview pane.

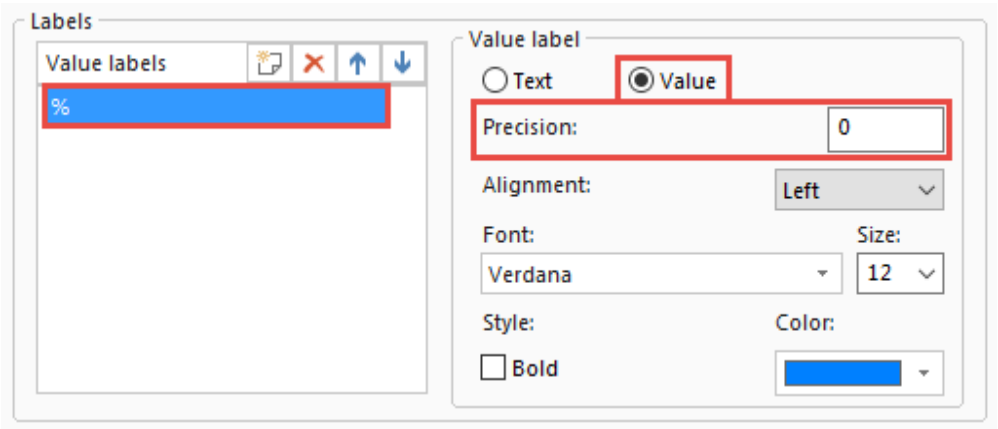
Labels

To create a value label, specify first whether the display element should show the current value (***Value***) or a text in place of the value (***Text***).

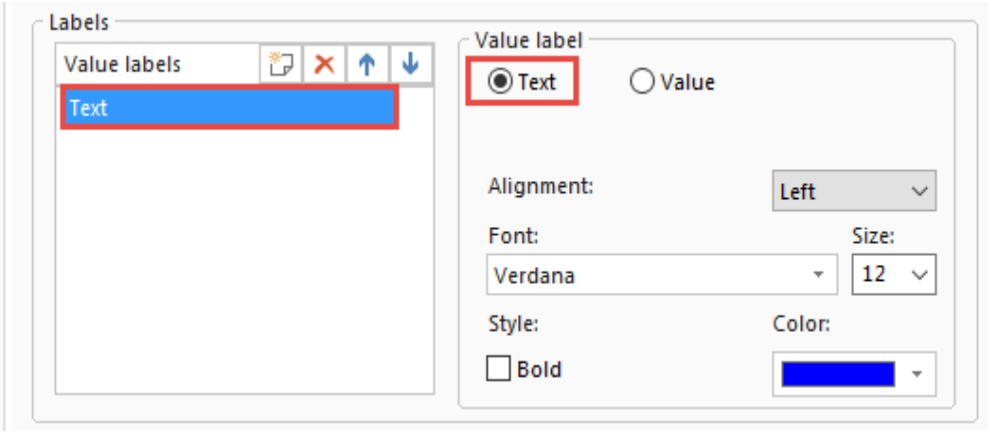


Value: If you want the display element to show the current value, select the radio button ***Value*** and click the button to create the value label. An input field opens where you can enter the unit. After specifying the unit, press the enter key.

Precision: Specify the number of significant digits.



Text: If the current value is to be replaced by a text, select the radio button ***Text*** and click the button to create the value label. An input field opens where you can enter the text. After entering the text, press the enter key.



Alignment: The value label can be displayed left-aligned, right-aligned or centered. *Default: Left*.

Font: Select the desired font and font size.

Style: The label font can be displayed in bold style. *Default: Disabled*.

Color: Select from the dropdown list a color for the label font.

The value labels can be moved up or down in the label list by clicking the arrow buttons . To delete a value label, click the button .

Important: Note that the value labels are listed in descending order: the lowest value is listed at the first position and the highest value at the last position.

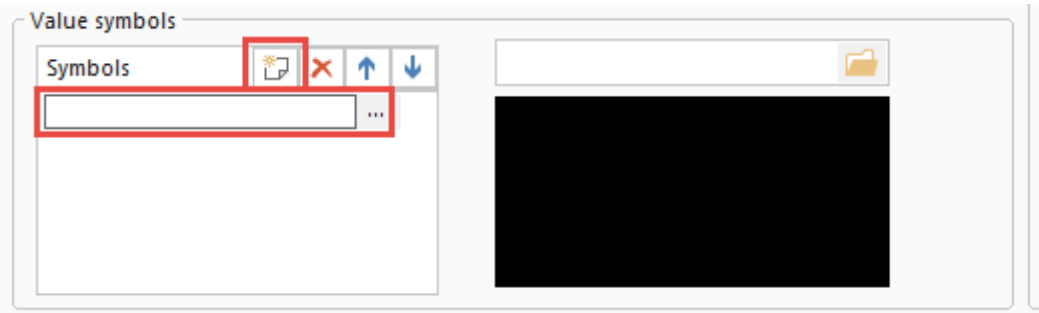
Value symbols

You can assign symbols to the values. In doing so, make sure that the order of the symbols matches the order of the value labels: the symbol for the lowest value has to be put at the first position and the symbol for the highest value at the last position. The intermediate values are interpolated.

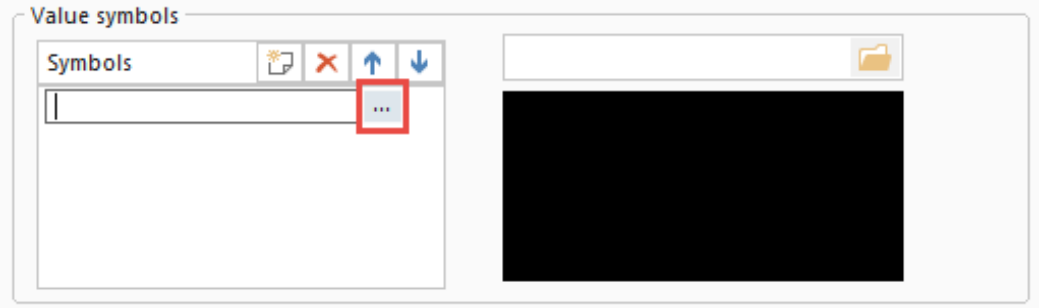
The position of the value symbols is important for the correct assignment of symbols to value labels!

Value symbols

To assign a symbol to a value, click the button . An input field opens where you can enter the name of the image file.



To browse the default directory for the desired image, click the browse button next to the input field.

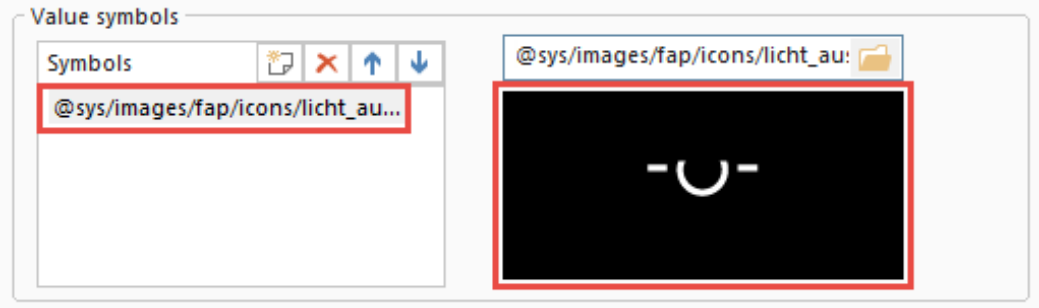


The dialog ***Open*** opens. Select the desired image and click ***Open***.

The selected image is displayed in the preview pane.

To replace a loaded image by another image, select the image you wish to replace and click the browse button above the preview pane.

The dialog ***Open*** opens. Select the desired image and click ***Open.***
The previously selected image is replaced by the newly selected image.



The value symbols can be moved up or down in the label list by clicking the arrow buttons . To delete a symbol, click the button .

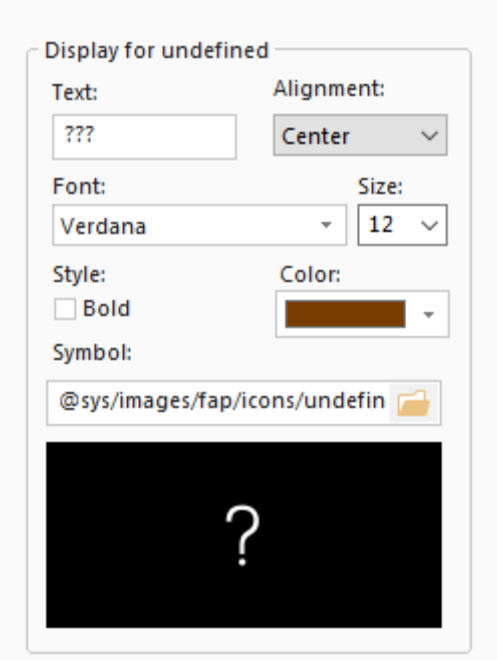
Important: Note that the order of the value labels must match the order of the symbols, so that the symbols can be assigned correctly to the value labels.

The first symbol in the list of the value symbols is assigned to the first value label (lowest value). The last symbol is assigned to the last value label (highest value). The intermediate values are interpolated.

Display for undefined

Here you can define a value label and a symbol which will be displayed when the value is unknown.

Display for undefined



Here you can define the display for the undefined value.

Text: Enter the text to be displayed when the value is unknown. *Default: ???*.

Alignment: The value label can be displayed left-aligned, right-aligned or centered. *Default: Centered*.

Font: Select the desired font and font size.

Style: The label font can be displayed in bold style. *Default: Disabled*.

Color: Select a font color from the dropdown list. *Default: brown*.

Symbol: Select a symbol for the unknown value: Enter in the input field the name of the image file or click the browse button and select the desired image from the default directory. The selected symbol is displayed in the preview pane.

5.2.1.5 Tab *Connections*

The tab ***Connections*** shows all process variables connected to the display element. In the column ***Function*** you can assign functions to the process variables.

Important: A context menu appears to set the functions directly when connecting display elements with process variables. If there is a definite assignment because just one function is possible, no context menu opens and the function is set automatically.

If more than one display element shall be connected with a process variable, the context menu only appears in these cases where the display elements have the same functions.
If they do not have the same functions, the variable will still be connected but ***not assigned to a function***.

Take care that you assign reasonable functions to the process variables before starting the visualization project in the server.

Disconnect:

You can disconnect process variables from the display element: select the desired process variables in the window ***Process variables*** and click the button ***Disconnect***.

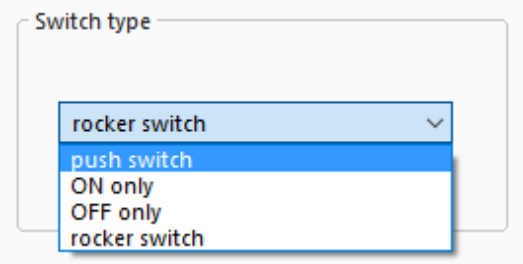
5.2.2 Control Properties

5.2.2.1 Switch

TAB *SWITCH*

On the tab ***Switch*** you can choose the switch type. The switch type determines the value that is sent to the bus when the display element or the slider are actuated.

Switch type



push switch: By clicking any point on the display element or slider, the value **1** or **0** is sent.

ON only: By clicking any point on the display element or slider, the value **1** is sent.

OFF only: By clicking any point on the display element or slider, the value **0** is sent.

rocker switch: By clicking the upper part of the display element or slider, the value **1** is sent. By clicking the lower part of the display element or slider, the value **0** is sent.

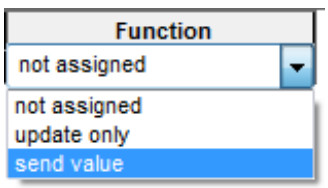
Default: rocker switch

TAB *CONNECTIONS*

The tab ***Connections*** shows all process variables connected to the display element *switch*.

Here you can assign functions to the connected process variables. To assign a function to a process variable, click in the column ***Function*** and select in the dropdown list the function you wish to assign to the process variable.

Function



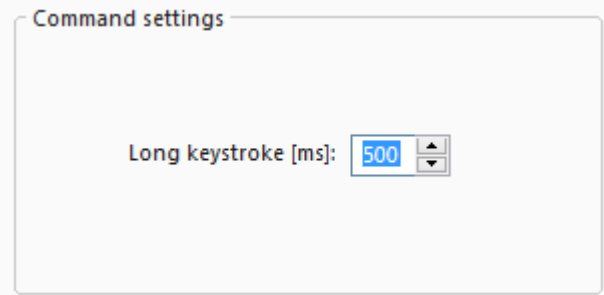
update only: Shows the updated value.

send value: Sends the value **1** or **0**.

5.2.2.2 Blind

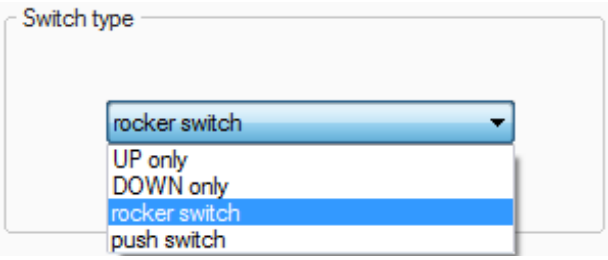
TAB *BLIND*

Command settings



On the tab ***Blind*** you can define the time interval for the long keystroke. The definition of the long keystroke is necessary for raising and lowering the blinds. Specify the time in **milliseconds**.

Switch type



UP only: By clicking on the display element or slider at any point, the blind is raised.
DOWN only: By clicking on the display element or slider at any point, the blind is lowered.

rocker switch: By clicking on the display element or slider in the upper area the blind is raised and by clicking in the lower area the blind is lowered.

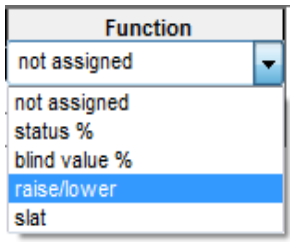
push switch: By clicking on the display element or slider at any point, the blind is raised or lowered.

Default: rocker switch

TAB *CONNECTIONS*

On the tab ***Connections*** you can assign functions to the process variables connected to the display element.

Function



status %: Indicates the current position of the blind.

blind value %: Values are sent to the bus when the slider handle is moved up or down.

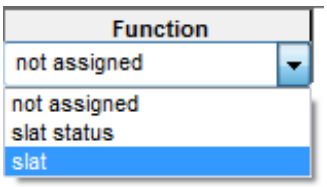
raise/lower: On long keystroke the blind is lowered or raised stepwise. The time interval for the long keystroke is defined on the tab ***Blind***.

slat: On short keystroke the slat adjustment is activated.

5.2.2.3 Slat

TAB *CONNECTIONS*

Function



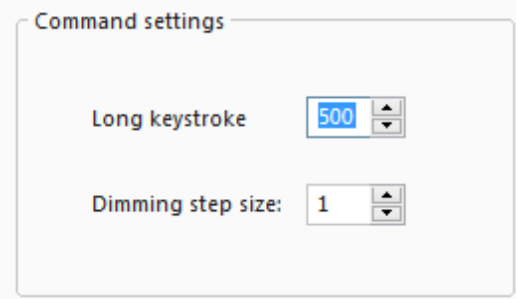
Slat status: Indicates the current position of the slats.

Slat: Sends values to the bus to adjust the slat position.

5.2.2.4 Dimmer

TAB *DIMMER*

Command settings



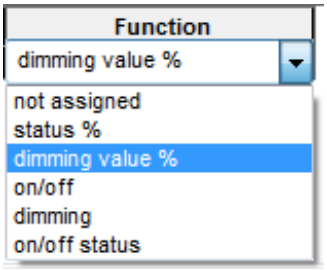
On the tab ***Dimmer*** you can define the time interval for the long keystroke.

The definition of the **Long keystroke** is necessary for sending dimming commands: on long keystroke in the upper part of the display element or slider, the light intensity increases. On long keystroke in the lower part of the display element or slider, the light intensity decreases. Specify the time in **milliseconds**.

Dimming step size: Set the step size for increasing or decreasing the light intensity stepwise.

TAB *CONNECTIONS*

Function



status %: Indicates the current value in percent.

dimming value %: Values for increasing or decreasing the light intensity are sent to the bus when the slider handle is moved up or down.

on/off: On short keystroke in the upper part of the display element or slider, the dimmer is switched on. On short keystroke in the lower part of the display element or slider, the dimmer is switched off.

dimming: On long keystroke in the upper part of the display element or slider, the light intensity is increased stepwise. On long keystroke in the lower part of the display element or slider, the light intensity is decreased stepwise.

The long keystroke and the step size can be parametrized on the tab ***Dimmer***.

on/off status: Indicates whether the dimmer is switched on or off. If the value is higher than 0%, the display element shows ON. If the value is 0%, the display element shows OFF.

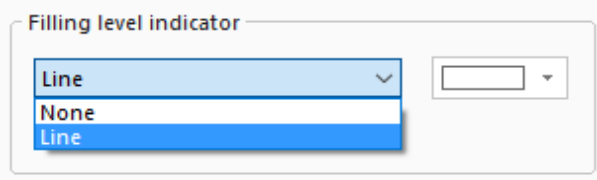
5.2.2.5 Multipurpose Control

The ***Multipurpose control*** can be used for different purposes. It can display as well as send values.

TAB *SLIDER*

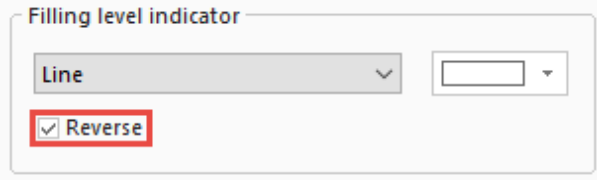
On the tab ***Slider*** you can define the look of the slider for the operation of the multipurpose control.

Filling level indicator



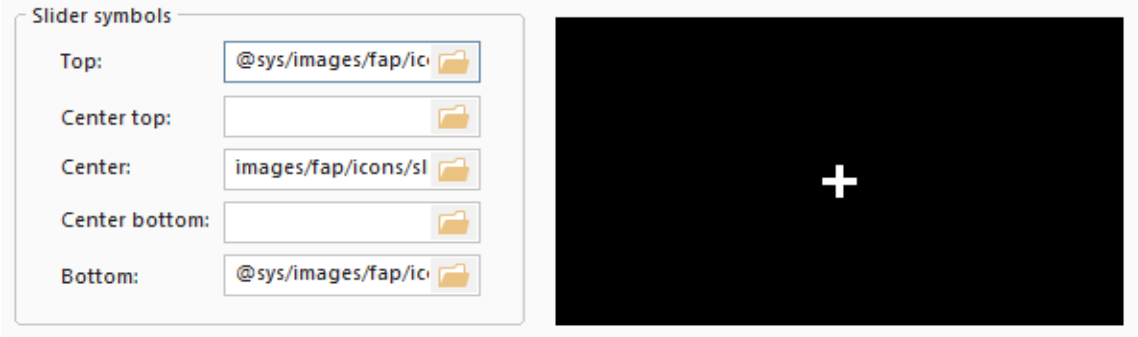
The increasing of values can be highlighted by a filling level indicator. To enable the filling level indicator, select the option ***Line*** in the dropdown list. You can select a color for the line in the dropdown list.

Reverse



Select the checkbox ***Reverse*** to invert the value scale of the slider: the highest value is at the bottom and the lowest value at the top of the scale.

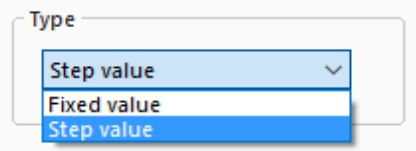
Slider symbols



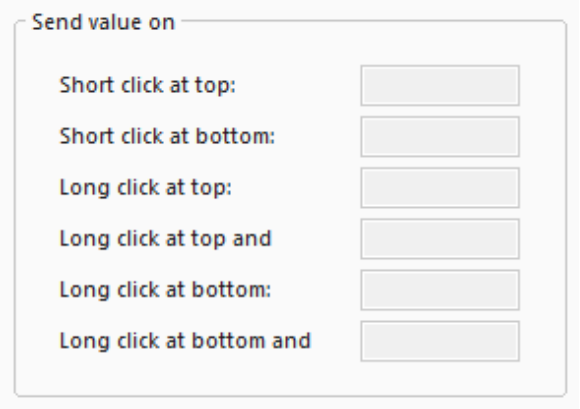
You can add symbols to the slider which are displayed at the top, center top, center, center bottom and bottom of the slider. To load an image, click the browse button. Select an image and click ***Open***. The selected image is shown in the preview pane.

TAB *MULTIPURPOSE CONTROL*

Type



Specify the value type to be sent to the bus.
Fixed value: The fixed value must be defined by the user. If this type is selected, you can specify under ***Send value on*** the values to be sent on certain mouse events.



Step value: The *step value* is calculated from the current value and the unit by which the current value is to be increased or decreased.

If you select this value type, you can specify under ***Step size*** the unit by which the current value is to be increased or decreased when the display element or the slider are actuated.

Send value on

If you have selected the value type ***fixed value***, you can enter here the values which are to be sent on certain mouse events.

Short click at top: Enter the value to be sent on a short click in the upper part of the slider or display element.

Short click at bottom: Enter the value to be sent on a short click in the lower part of the slider or display element.

Long click at top: Enter the value to be sent on a long click in the upper part of the slider or display element.

Long click at top and release: Enter the value to be sent when the display element or slider is released after a long click in the upper part.

Long click at bottom: Enter the value to be sent on a long click in the lower part of the display element or slider.

Long click at bottom and release: Enter the value to be sent when the display element or slider is released after a long click in the lower part.

Command settings

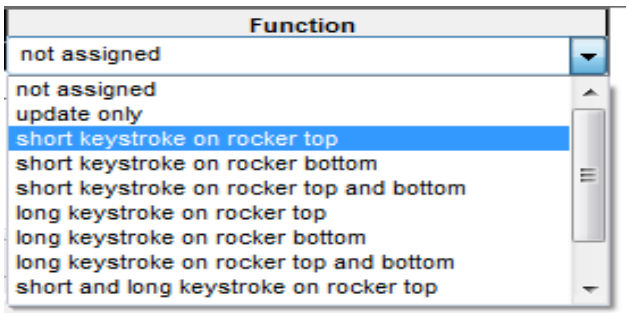
Specify the time interval for the long keystroke. This entry is necessary if you assign a function with a long keystroke to a process variable on the tab ***Connections***.

Step size

If you select the value type step value, you can specify here the unit by which the current value is to be increased or decreased when the display element or the slider are actuated.

TAB *CONNECTIONS*

Function



On the tab ***Connections*** you can assign functions to the connected process variables.

5.2.2.6 Control Send Fixed Value & Control Step Value

The elements **Control send fixed value** and **Control step value** have the same functionality as the multipurpose control with his appropriate setting.

5.2.2.7 Slider

The slider exists in two versions, one in horizontal and in a vertical representation.

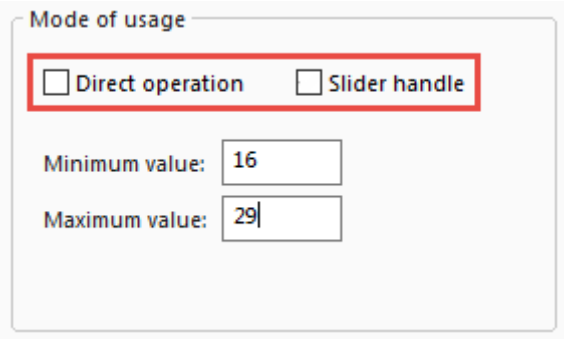
The slider serves the purpose of operating display elements. There are two ways to use the slider:

- direct operation: values are sent by clicking on the slider
- operation via slider handle: values are sent by moving the slider handle up or down.

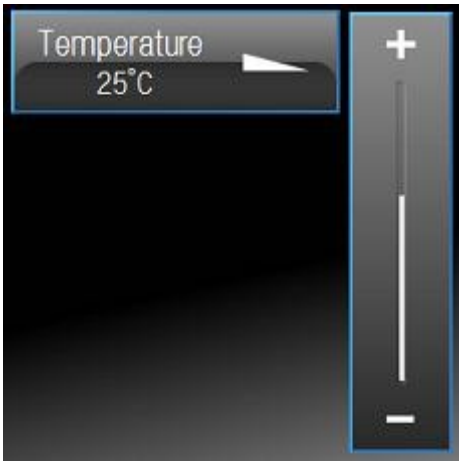
You can specify the mode of usage of the display element on the tab ***General*** in the property dialog of the element.

Direct operation

To control a display element via slider, clear the checkbox ***Direct operation*** and ***Slider handle***.

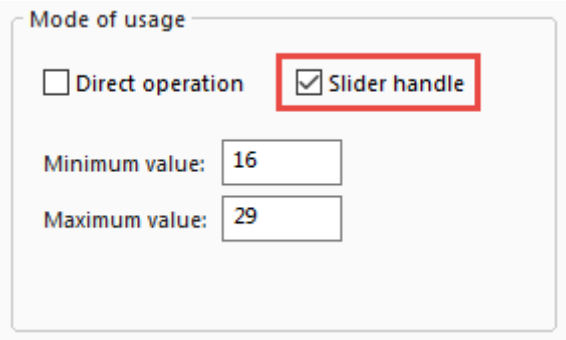


When the element is selected in the server, the slider is activated and values can be sent by clicking the slider.

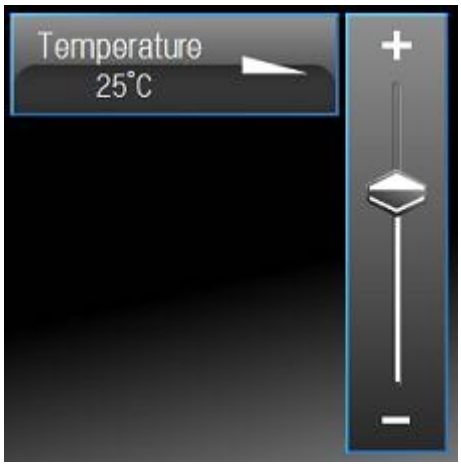


Operation via slider handle

To operate a display element via slider handle, select the checkbox ***Slider handle***.

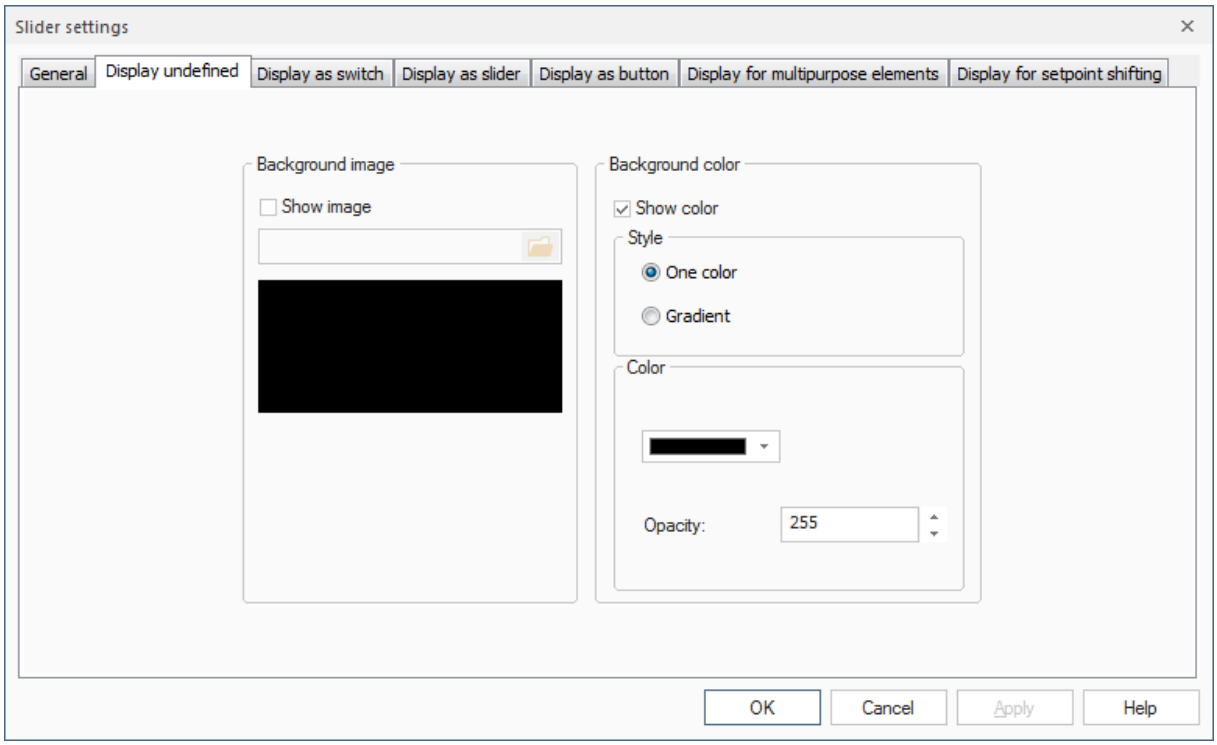


When the element is selected in the server, the slider with the slider handle is activated and values can be sent by moving the slider handle up or down.



A slider should be contained in a worksheet only once. It always applies to the currently selected display element. The appearance of the slider depends on the selected display element.

TAB *DISPLAY UNDEFINED*



The tab ***Display undefined*** contains the display configuration for the undefined state of the slider. It is possible to configure the background image and background color.

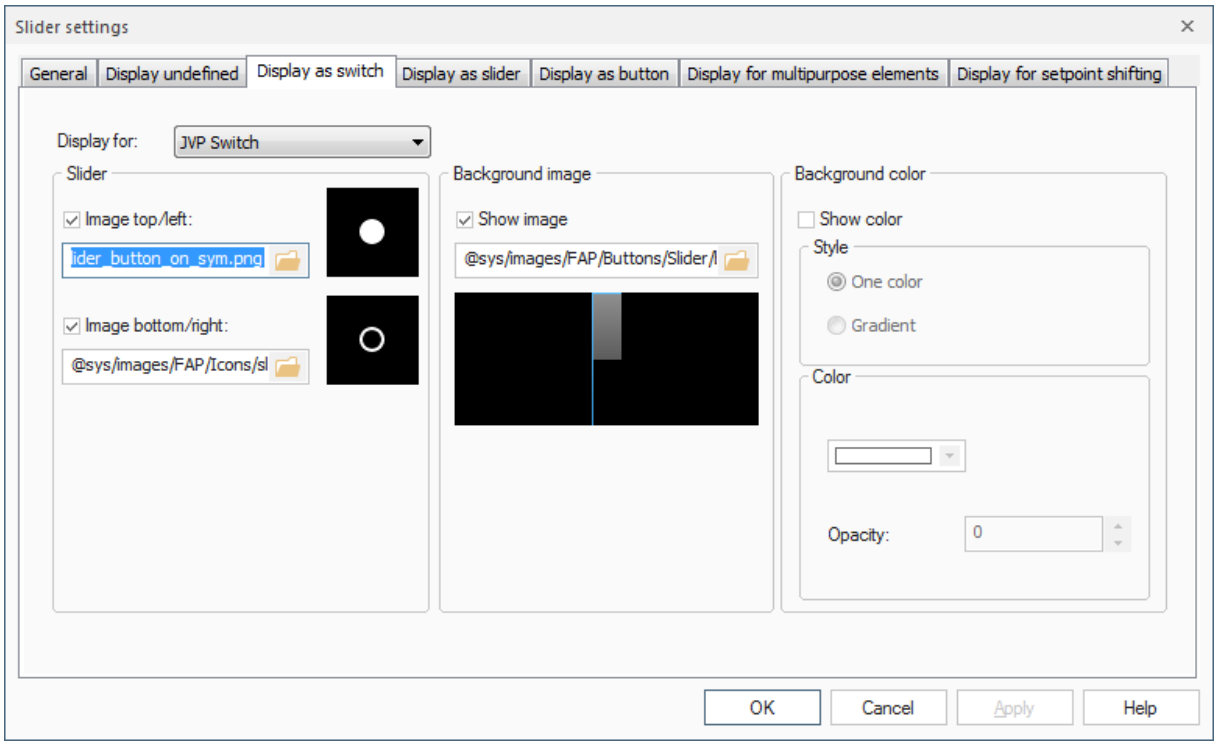
Background color: You can choose the background color by enabling the check box and choosing a color in the dialog. If a background image is chosen, it will be on top of the background color.

There is the possibility to choose between one color background with opacity or a color gradient.

Background image: You can choose a background image by enabling the check box and providing a path to the image you want to use.

If **nothing** is activated, the element will be transparent.

TAB *DISPLAY AS SWITCH*



The tab ***Display as switch*** contains the display configuration for all elements which use the slider as a switch.

Following elements can be chosen to be configured:

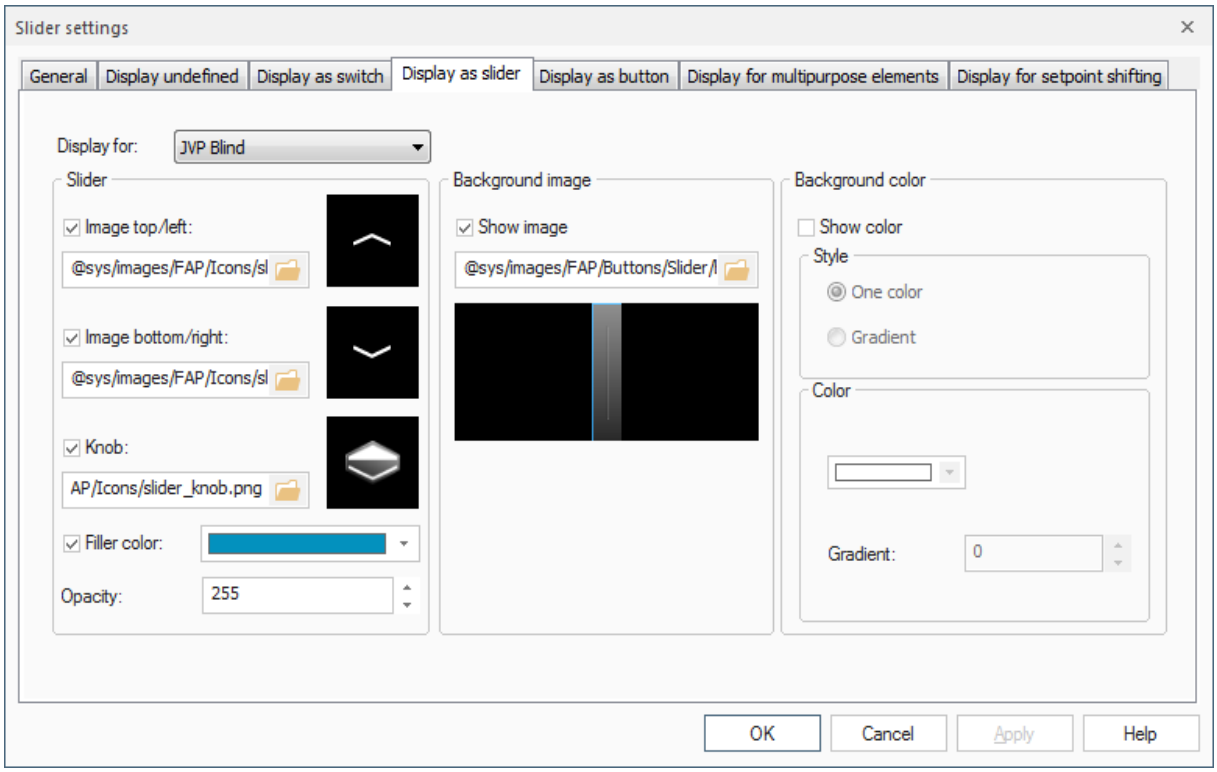
- Switch
- Colour control
- Knob
- Dimmer knob
- Tuneable white control

Following options can be configured, alongside the background color and background image:

Image top/left: You can choose the icon, which should be displayed in the upper area (vertical slider) or the left area (horizontal slider) respectively.

Image bottom/right: You can choose the icon, which should be displayed in the lower area (vertical slider) or the right area (horizontal slider) respectively.

TAB *DISPLAY AS SLIDER*



The tab ***Display as slider*** contains the display configuration for all elements which use the JUNG slider as a slide control.

Following elements can be chosen to be configured:

- Blind
- Dimmer
- Slat

Following options can be configured, alongside the background color and background image:

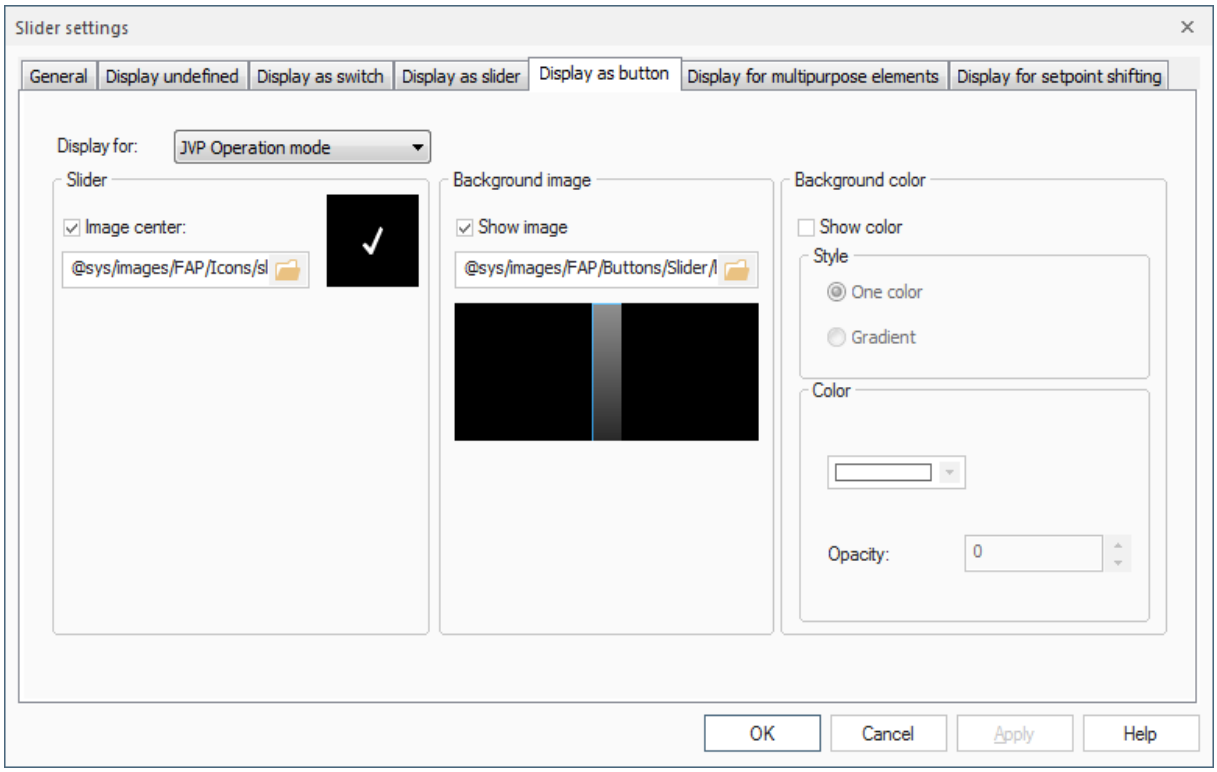
Image top/left: You can choose the icon, which should be displayed in the upper area (vertical slider) or the left area (horizontal slider) respectively.

Image bottom/right: You can choose the icon, which should be displayed in the lower area (vertical slider) or the right area (horizontal slider) respectively.

Knob: You can choose the icon, which will be displayed as the knob of the slide control.

Filler color: You can choose a color and an opacity for the slide control. The color will be displayed as the fill level in the slide control, if the knob is moved.

TAB DISPLAY AS BUTTON



The tab ***Display as button*** contains the display configuration for all elements which use the JUNG slider as a button.

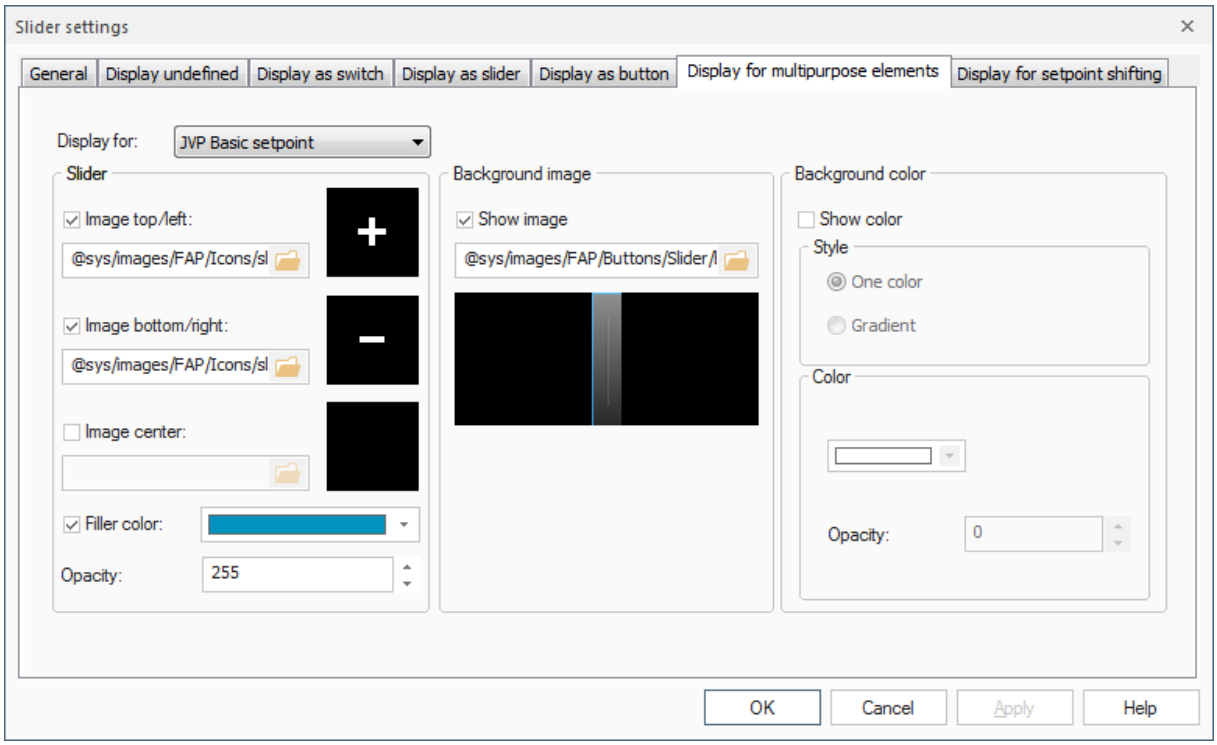
Following elements can be chosen to be configured:

- Operation mode
- Operation mode switch over

Following options can be configured, alongside the background color and background image:

Image center: You can choose the icon, which will be displayed in the middle of the slider.

TAB DISPLAY FOR MULTIPURPOSE ELEMENTS



The tab ***Display for multipurpose elements*** contains the display configuration for all elements which are mainly used to only display values.

Following elements can be chosen to be configured:

- Basic setpoint
- Multipurpose control
- Control send fixed value
- Control step value

Following options can be configured, alongside the background color and background image:

Image top/left: You can choose the icon, which should be displayed in the upper area (vertical slider) or the left area (horizontal slider) respectively.

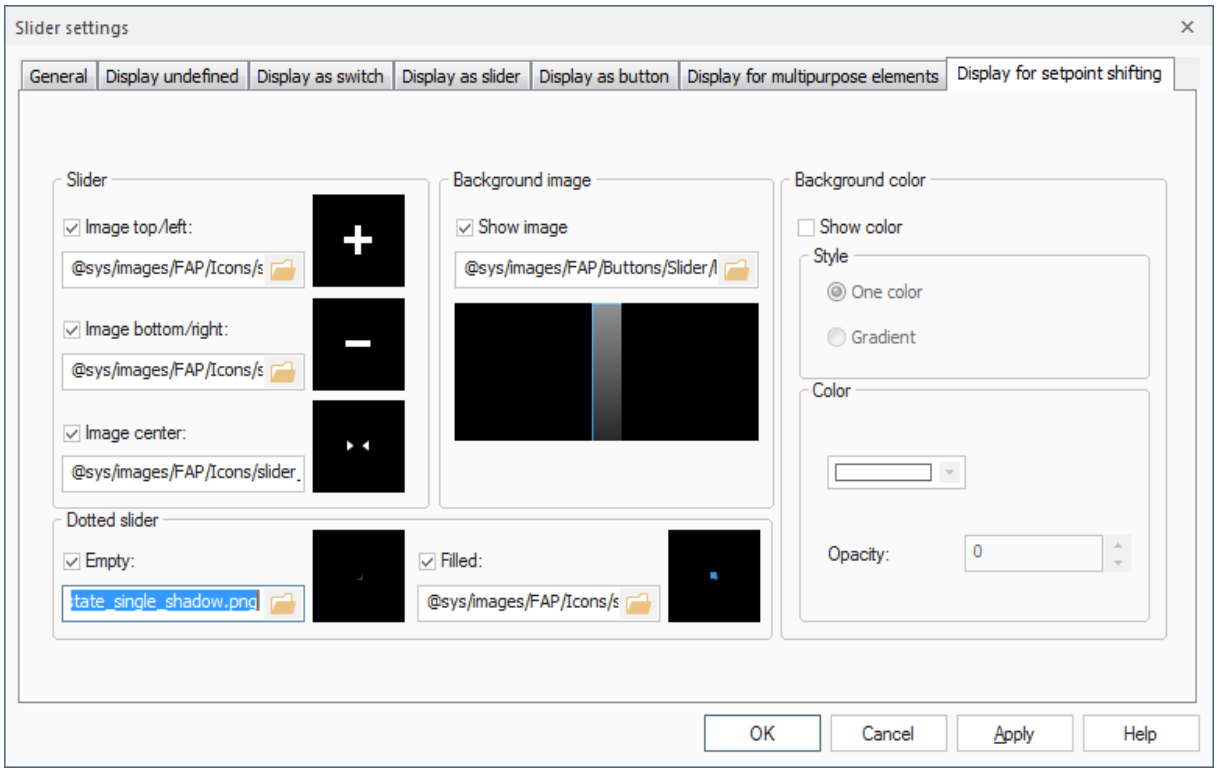
Image bottom/right: You can choose the icon, which should be displayed in the lower area (vertical slider) or the right area (horizontal slider) respectively.

Image center: You can choose the icon, which will be displayed in the middle of the slider.

Filler color: You can choose a color and an opacity for the slide control. The color will be displayed as the fill level in the slide control.

Note: ***Multipurpose control, Control send fixed value*** *and* ***Control step value*** offer further individual settings for the slider. More details can be found in chapter 5.2.2.5 Multipurpose Control under the tab ***Slider***.

TAB DISPLAY FOR SETPOINT SHIFTING



The tab ***Display for setpoint shifting*** contains the display configuration for the element setpoint shifting.

Following options can be configured, alongside the background color and background image:

Image top/left: You can choose the icon, which should be displayed in the upper area (vertical slider) or the left area (horizontal slider) respectively.

Image bottom/right: You can choose the icon, which should be displayed in the lower area (vertical slider) or the right area (horizontal slider) respectively.

Image center: You can choose the icon, which will be displayed in the middle of the slider.

Note: The setpoint shifting can be set in multiple steps. Every step will be displayed as a dot on the slider. Therefor you can set icons for filled and empty dots.

Dotted slider empty: You can choose the icon, which will be displayed as an empty dot on the slider.

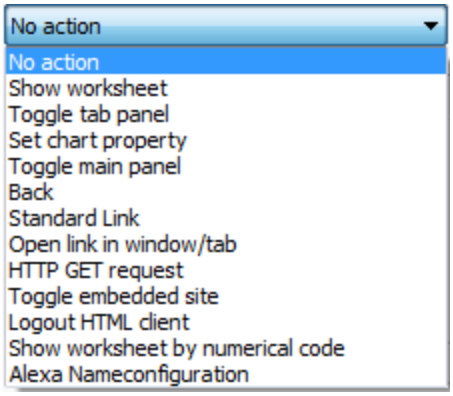
Dotted slider filled: You can choose the icon, which will be displayed as a filled dot on the slider.

5.2.2.8 Button

The display element *button* serves to initiate certain actions within the visualization itself, such as toggling worksheets. It is not connected to process variables.

TAB *ACTION*

On the tab ***Action*** you can select the action to be initiated when the display element is actuated.

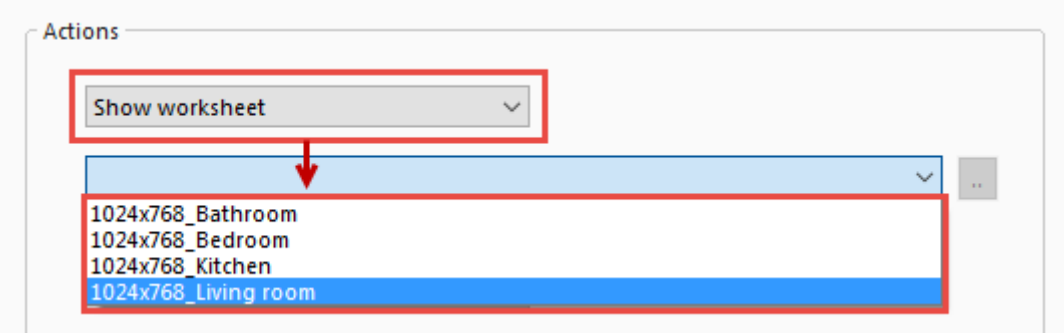


No action: The display element does not initiate any action when clicked.

5.2.2.8.1 Show Worksheet

A worksheet is opened when the display element is clicked.

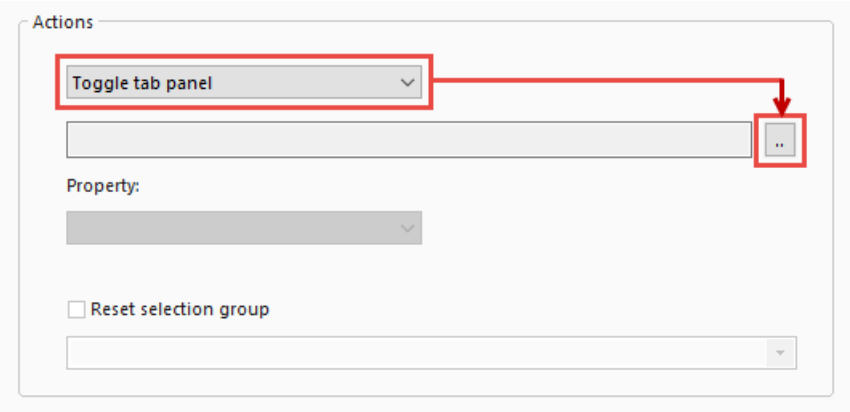
Choose the action ***Show worksheet*** and select from the dropdown list below the worksheet you want to be opened when the button is actuated.



5.2.2.8.2 Toggle Tab Panel

A tab in a tab panel is activated when the button is clicked.

Choose the action ***Toggle tab panel*** and click the browse button below.



The dialog ***Select tab*** opens. It shows all tab panels inserted in the current worksheet. Select the tab you want to be activated when the button is clicked and click the button ***OK***.

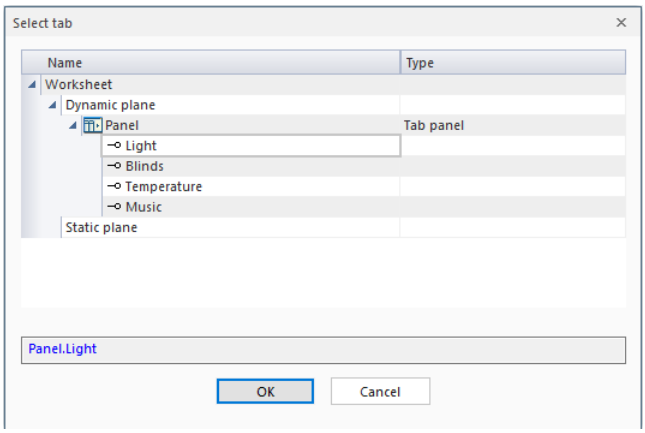
The path of the tab is shown in the input field below the dropdown list.

Note:

The path changes if the display element *button* is inserted into a panel or if the name of the panel in which the button resides, has been changed. If the entered path of the tab is not consistent with the actual path, the action ***Toggle tab panel*** cannot be initiated. In this case the path must be **updated**!

5.2.2.8.3 Set Chart Property

Sets a property in a consumption data chart. Select the action ***Set chart property*** and click the browse button.



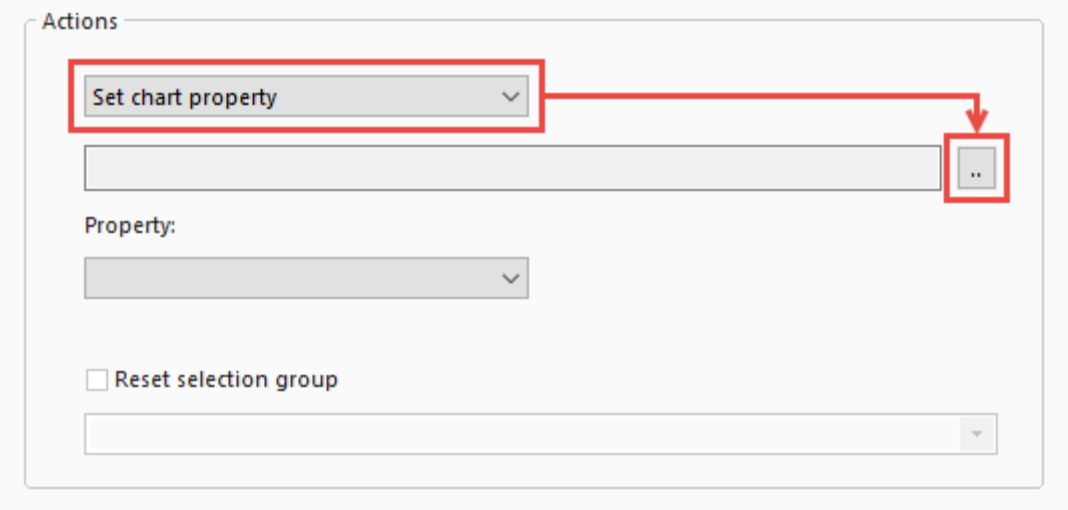
The dialog ***Select chart*** opens. Select the chart in which to set a property and click ***OK***.

The path of the chart is shown in the input field below the dropdown list.

Note:

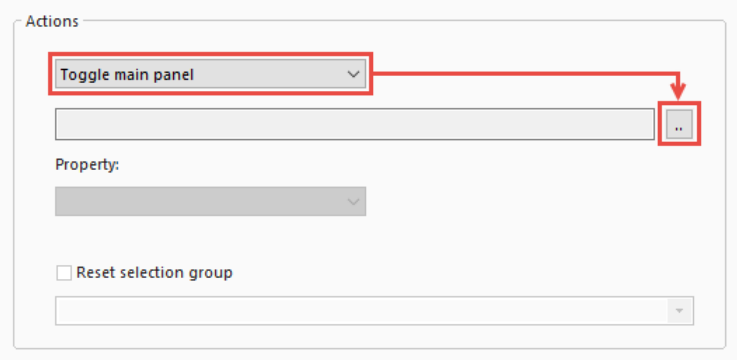
The path changes if the display element *button* is inserted into a panel or if the name of the panel in which the button resides, has been changed. If the entered path of the chart is not consistent with the actual path, the action ***Set chart property*** cannot be initiated. In this case the path must be **updated**!

Select under ***Property*** the property to be set when the button is clicked.



5.2.2.8.4 Toggle Main Panel

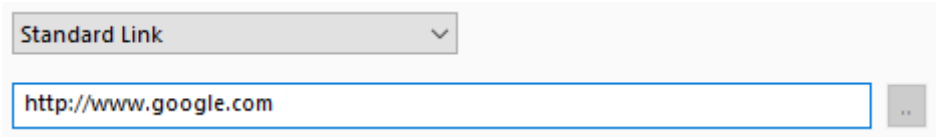
Initiates the rotation of the main panel. Select the action ***Toggle main panel*** and click the browse button.



The dialog ***Select main panel*** opens.

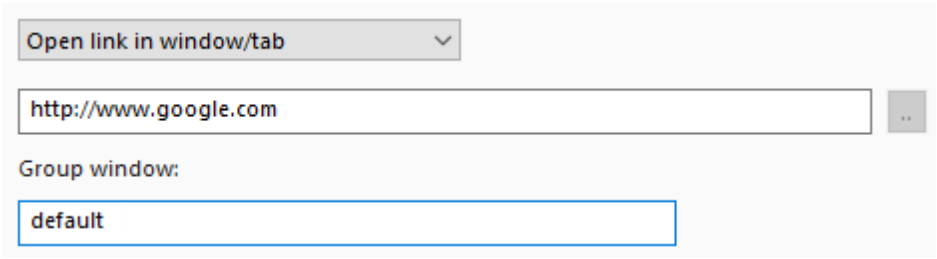
Select the main panel to be rotated and click ***OK***.

5.2.2.8.5 Standard Link



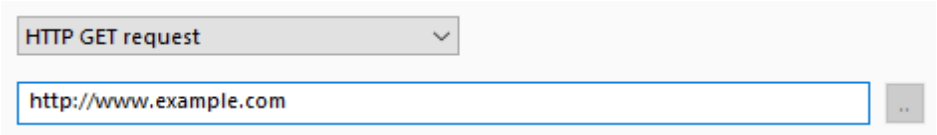
The Button behaves like a normal link. The specified, linked page will be loaded by click. The entered URL will not be tested for the correct phrasing and the behavior after the click depends on your browser.

5.2.2.8.6 Open Link in Window/Tab



Extended form of the *Standard Link*. With this control, it is possible to open a link in a new browser/tab. To prevent from opening windows endlessly, there is the option to enter a name for the ***group window***. All link buttons that include this group name will be loaded in the same window.
The name of the group window can not start with an underscore.

5.2.2.8.7 HTTP GET Request

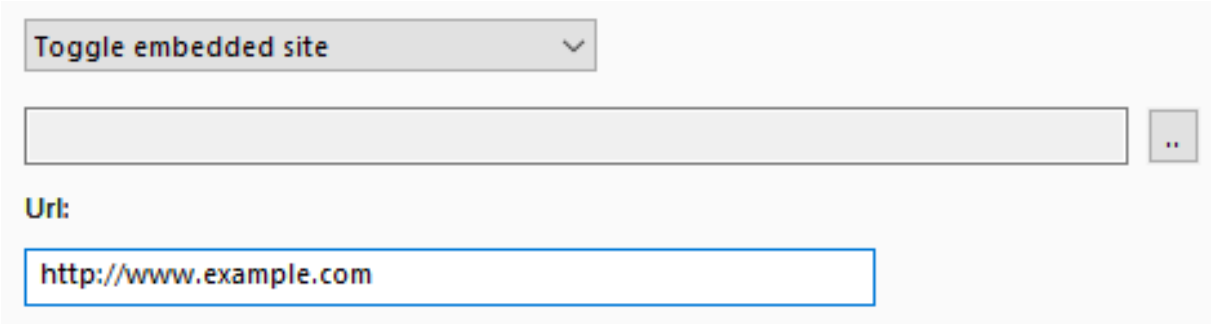


It will not open a new window/tab. Suitable for updating cameras, for example.
The entered URL will not be tested for the correct phrasing and the behavior after the click depends on your browser.

5.2.2.8.8 Back

Go to previous worksheet, if there is one. This is possible with up to 100 jumps.

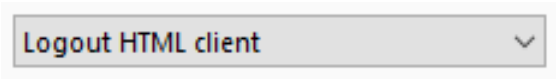
5.2.2.8.9 Toggle Embedded Site



After the element whose content will be changed is selected, you can enter a URL to be called when pushing the button. For the link the same rules, as noted under item 5.1.18, count.

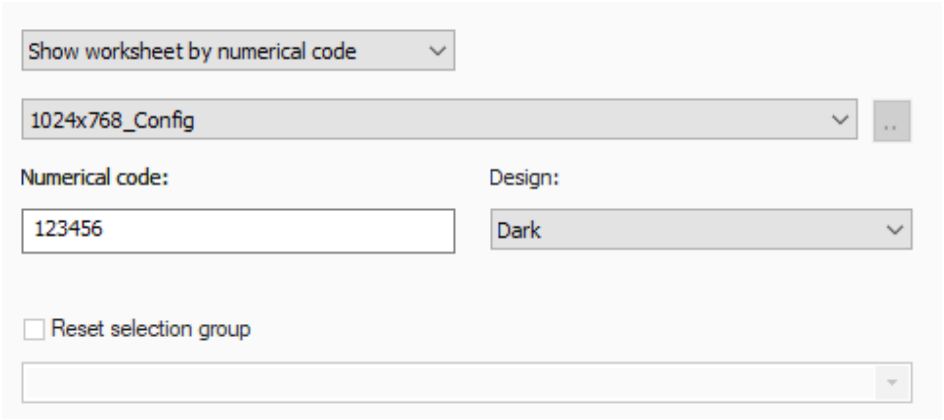
NOTE: The address of the external page must have the prefix **http://** to make sure, that the address can be resolved correctly.

5.2.2.8.10 Logout HTML Client



By clicking a button with these setting, the login page of the server will be loaded.

5.2.2.8.11 Show Worksheet by Numerical Code

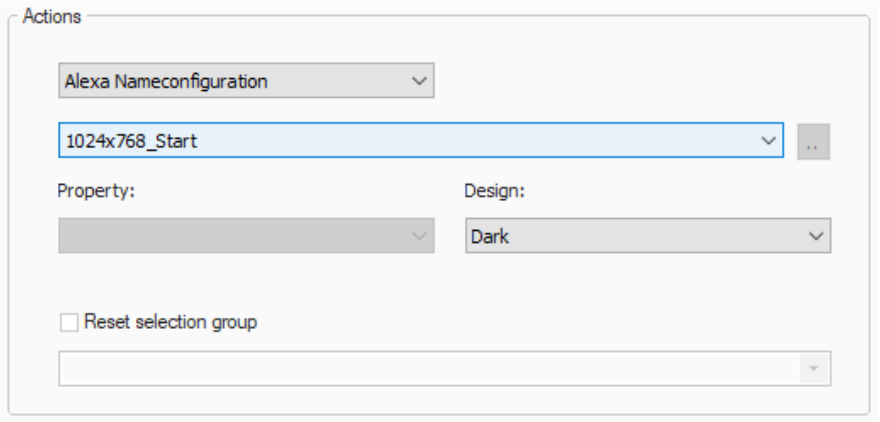


This action has the same effect as ***show worksheet***: when pressing the button the selected worksheet will open. The difference lies in the definition of a numerical code. The new worksheet will only load if it has been entered correctly.

The ***Design*** option determines the color palette of the overlay, which is displayed in the server after pressing the button.

5.2.2.8.12 Alexa Nameconfiguration

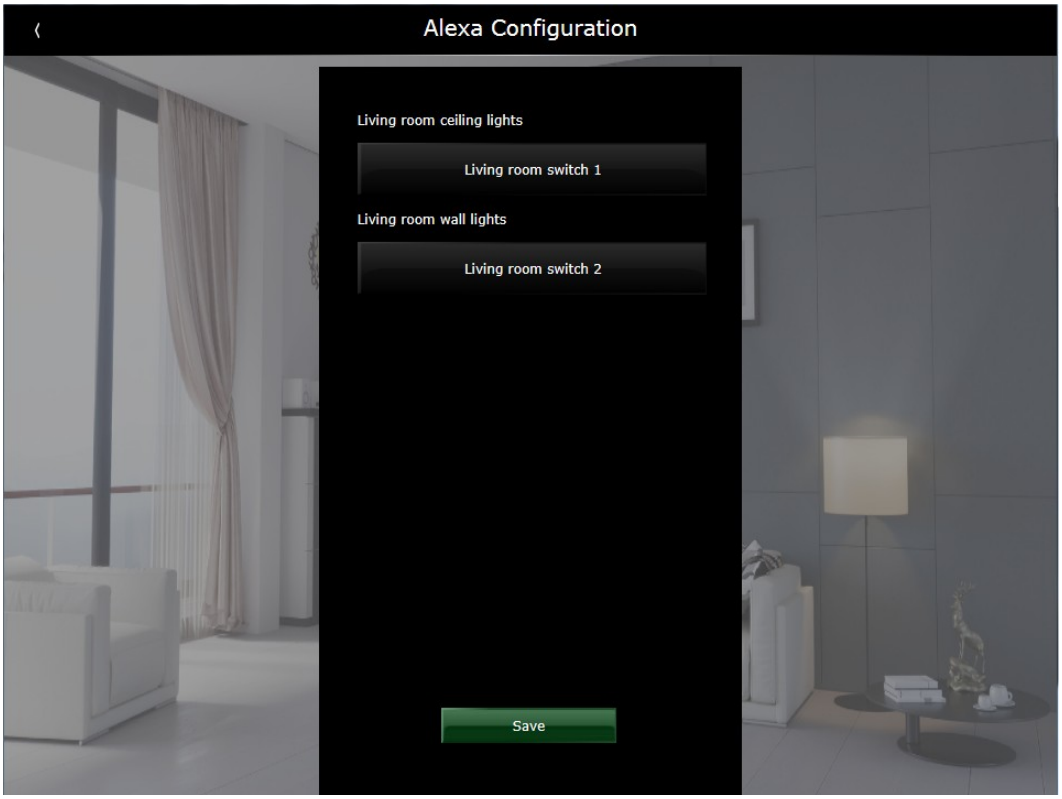
This action opens the Alexa Nameconfiguration in the visualization server. This overlay displays all the data points that are on the selected page and are activated for the Alexa voice control. If you want a general survey of all data points, you should select <All Worksheets> instead of a specific worksheet.



The ***Design*** option determines the color palette of the overlay, which is displayed in the server after pressing the button.

Operation in the HTML overlay

A label with the data point description and a text field are displayed in the overlay for each data point on the selected worksheet, which has been activated for the Alexa voice control. To change the Alexa name of a data point, enter the new name in the respective text field and confirm with Save. A new search of the devices via the Alexa app should now find the data point with the updated name.

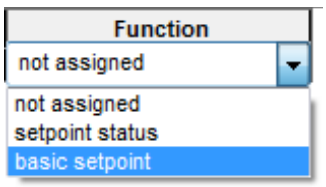


5.2.2.9 Basic Setpoint

TAB *CONNECTIONS2*

On the tab ***Connections*** you can assign functions to the connected process variables.

Function



Setpoint status: Shows the status of the setpoint.

Basic setpoint: Sends the basic setpoint value to the bus.

5.2.2.10 Basic Setpoint Shifting

TAB *SETPOINT SHIFTING*

Note: The settings made on the tab *Basic setpoint shifting* apply to the devices of the company **JUNG** and must therefore comply with the parameters of these devices.

Device-specific settings

Number of steps: Specifies the range within which the setpoint can be shifted.

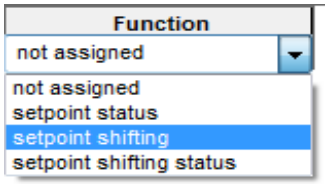
Step size in Kelvin: The step size must correspond to the step size specified for the controller in the ETS tool software.

Valence of a step in the controller: For the correct setting of the valence, see the description of the JUNG device.

TAB *CONNECTIONS*

Function

On the tab ***Connections*** you can assign functions to the connected process variables.



Setpoint status: Displays the setpoint status.

Setpoint shifting: Sends a shifting command when the slider or display element is actuated.

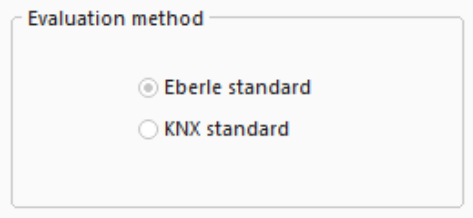
Setpoint shifting status: Displays the status of the setpoint shifting.

5.2.2.11 Operation Modes

The display elements **Night**, **Comfort**, **Standby** and **Frost protection** serve exclusively the purpose of controlling the room temperature controllers of the company **JUNG**. The settings made in the property dialogs of these elements must comply with the parametrization of the devices of the company **JUNG**!

TAB *OPERATION MODES*

Evaluation method



It is possible to choose either the Eberle standard or the KNX standard.

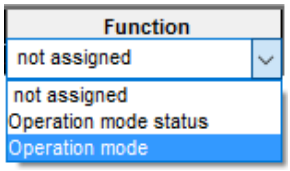
Eberle standard: Sends/receives the numbers 1, 2, 4 and 8 which are determined by the Eberle standard.

KNX standard: Sends/receives the numbers 1, 2, 3 and 4 which are determined by the KNX standard.

TAB *CONNECTIONS*

On the tab ***Connections*** you can assign functions to the connected process variables.

Function



Operation mode status: Displays the status of the operation mode (active, inactive or undefined).

Operation mode: The operation mode is activated when the display element or the slider are clicked.

5.2.2.12 Operation Mode Switch Over

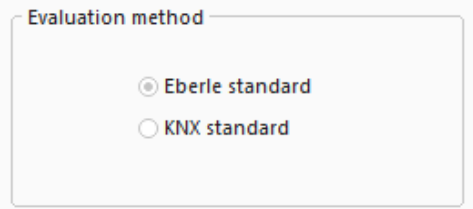
The operation mode switch over has the same functionality as the display elements **night**, **comfort**, **standby** and **antifreeze** (see previous point).

TABS UNDEFINED*, COMFORT, STANDBY, NIGHT, FROST PROTECTION*

On this tabs you can configure the display for the separate states of the operation modes (see chapter 5.2.1.3 *Tab Display*)

TAB *OPERATION MODE*

Evaluation method

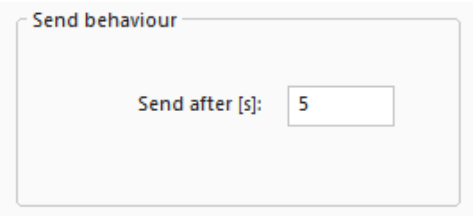


It is possible to choose either the Eberle standard or the KNX standard.

Eberle standard: Sends/receives the numbers 1, 2, 4 and 8 which are determined by the Eberle standard.

KNX standard: Sends/receives the numbers 1, 2, 3 and 4 which are determined by the KNX standard.

Send behavior

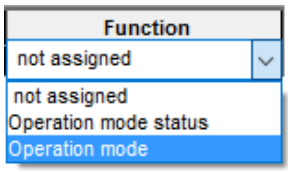


Here you can set the waiting period before the state is sent after pressing the control element.

TAB *CONNECTIONS*

On the tab ***Connections*** you can assign functions to the connected process variables.

Function



Operation mode status: Displays the status of the operation mode (active, inactive or undefined).

Operation mode: The operation mode is activated when the display element or the slider are clicked.

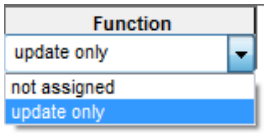
5.2.2.13 Multipurpose Display

The display element ***Multipurpose display*** serves to display values. It is not used to send commands to the bus.

TAB *CONNECTIONS*

On the tab ***Connections*** you can assign functions to the connected process variables. The display element ***Multipurpose display*** cannot send commands to the bus. It only displays values. Therefore only the function ***update only*** can be assigned to the connected process variables.

Function

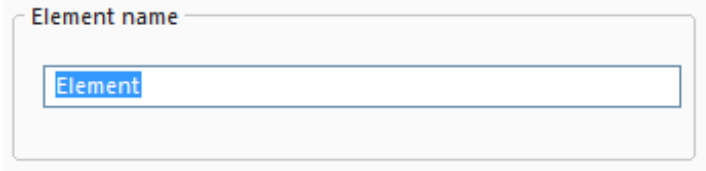


Update only: Shows the current value.

5.2.2.14 Image Display

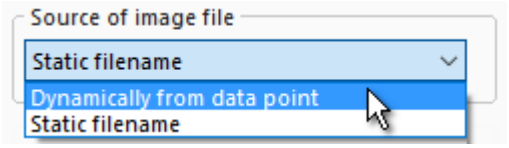
TAB *IMAGE DISPLAY*

Element name

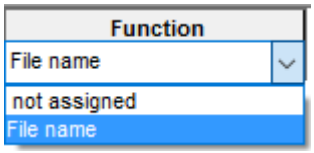


Enter an appropriate element name for the display element.

Dynamically from data point



If you select the source ***Dynamically from data point,*** the display element must be connected with a text variable of the type **string** which contains the name of the image file and the file extension (for example image.jpg).

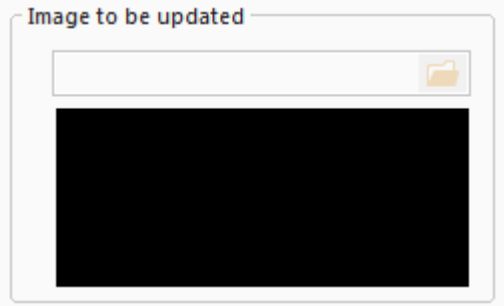


On the tab ***Connections*** assign to the connected process variable the function ***File name***.

Image for undefined status

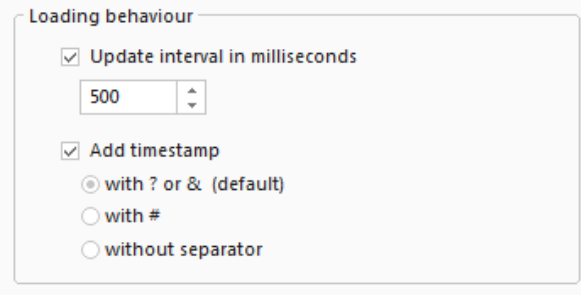
Select the image to be displayed if the specified image does not exist or the file name is incorrect.

Static filename



If you select the source ***Static filename***, enter an image under ***Image to be updated.*** Then specify the update interval for the image.

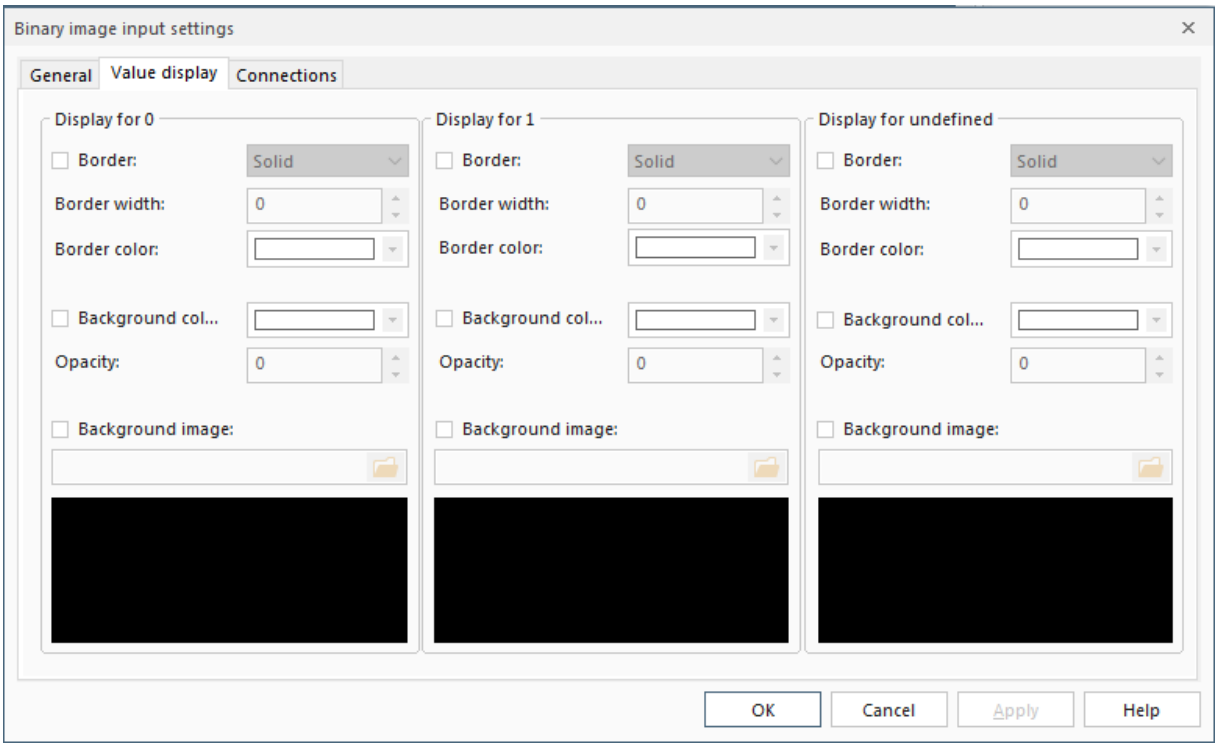
Update interval in milliseconds



Specify the update interval in milli-seconds.

Add timestamp: Sometimes it is necessary to attach a time stamp. Here you can configure with which character the timestamp will be attached to the name.

5.2.2.15 Binary Image Input



Display for 0	Path to the image, that is displayed when the value is 0.
Display for1	Path to the image, that is displayed when the value is 1.
Display for undefined	Path to the image, that is displayed when the value is undefined.

Supported file formats:

- Windows or OS/2 Bitmap (*.bmp)
- CompuServe Graphics Interchange (*.gif)
- JPEG (*.jpg)
- Portable Network Graphics (*.png)
- Tagged Image File Format (*.tif)

Note: Animated gifs are not supported.

Note: *For to ensure that all graphics of the project are integrated during export in the control panel, all paths are converted into symbol paths. That way the path specifications do not have to be reconfigured if the project has been imported to a different computer.*

@sys: *This prefix replaces the complete path to the design directory under '**<Installation directory>/App/VisSysResources/Design/**'.*
Graphics with a path from this directory structure remain at this location.

@proj: *This prefix replaces the complete path to the project directory under '**<Installation directory>/Stations/local/projects/Project name .VIS/images/**'*
Graphics with a path from this directory structure are already in the project directory*.*

*All other graphics to be displayed in the visualization are automatically copied into the image directory of the current project **<@ proj / images>**.*

5.2.2.16 Binary Image Output

A more specific description of the **Value display** can be found at **7.2.2.13 Binary Image Input.**

The binary image output can be configured as a switch (push switch, ON only, OFF only, rocker switch).

At the tab ***Connections***, for every connected process variable can be chosen if a value should be sent or the element should only be updated.

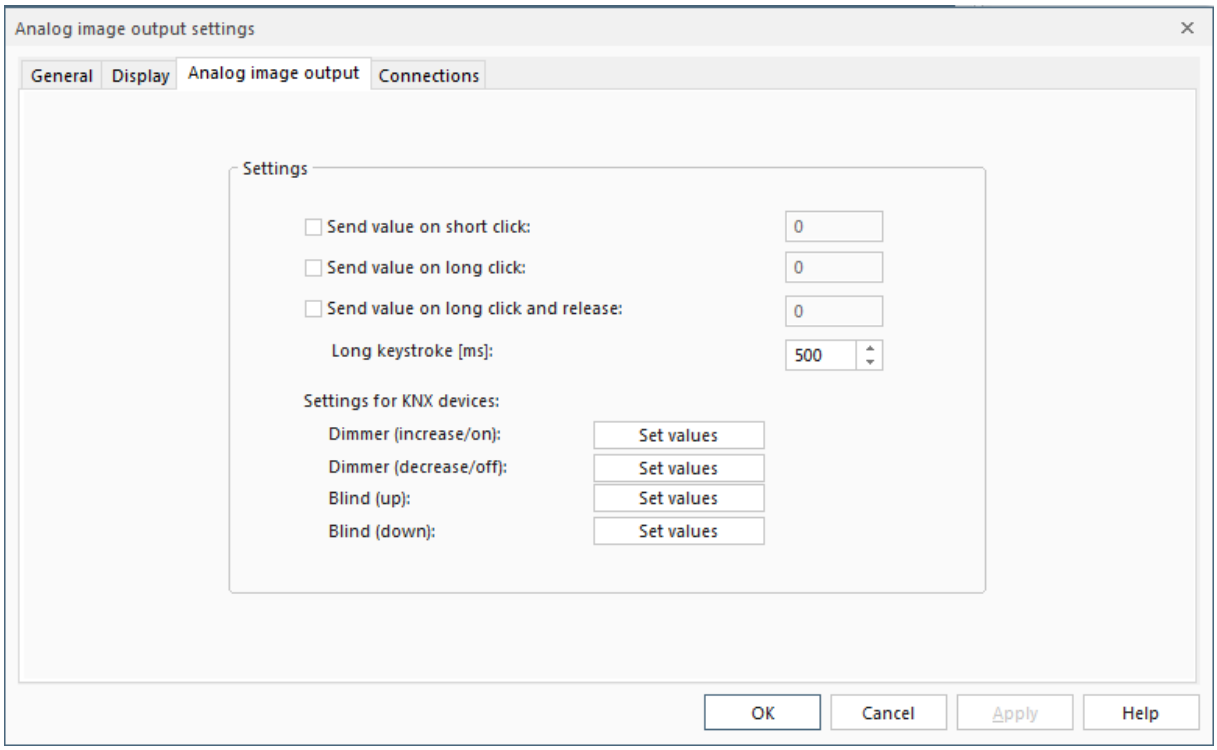
5.2.2.17 Analog Image Output

The analog image output can be used to send an analog value to the process. Therefor, different values can be assigned for “short click”, “long click” and “long click and release”.

Depending on the value settings special display elements can be created for example the KNX blind control or the KNX dimmer. The KNX specific value settings for these elements can be set in the configuration dialog.

The analog image output can be connected to both analog and binary process variables. This is required at the parametrization of the KNX dimmer for example.

At the tab ***Connections***, it can be configured for every connected process variable at which kind of click a value should be sent.



Send value on short click	Insert the value to be sent on short click. *Default: 0*.
Send value on long click	Insert the value to be sent on long click. *Default: 0*.
Send value on long click and release	Insert the value to be sent on release after a long click. *Default: 0*.
Long keystroke [ms]	Set the time interval for the long click. *Default: 250 ms*.
Dimmer (increase/on)	Set values: inserts the values for KNX
Dimmer (decrease/off)	Set values: inserts the values for KNX
Blind (up)	Set values: inserts the values for KNX
Blind (down)	Set values: inserts the values for KNX
Supported file formats	• Windows or OS/2 Bitmap (*.bmp) • CompuServe Graphics Interchange (*.gif) • JPEG (*.jpg) • Portable Network Graphics (*.png) • Tagged Image File Format (*.tif)

Note: Animated gifs are not supported.

Note: *For to ensure that all graphics of the project are integrated during export in the control panel, all paths are converted into symbol paths. That way the path specifications do not have to be reconfigured if the project has been imported to a different computer.*

@sys: *This prefix replaces the complete path to the design directory under '**<Installation directory>/App/VisSysResources/Design/**'.*
Graphics with a path from this directory structure remain at this location.

@proj: *This prefix replaces the complete path to the project directory under '**<Installation directory>/Stations/local/projects/Project name .VIS/images/**'*
Graphics with a path from this directory structure are already in the project directory*.*

*All other graphics to be displayed in the visualization are automatically copied into the image directory of the current project **<@ proj / images>**.*

5.2.2.18 Text Input

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to show incoming value changes.

5.2.2.19 Text Input/Output

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes or the function ***Send value*** to send entered values.

5.2.2.20 Numeric Value Input/Output

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes or the function ***Send value*** to send entered values.

5.2.2.21 Date/Time Input

On the tab ***Value display***, you can configure the display for the values "Defined" and "Undefined". You can define whether the date and/or time should be displayed for the element. To do so, activate the corresponding options ("Show date" and/or "Show time").

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes.

5.2.2.22 Timestamp

The timestamp shows **the last time** a connected data point **received** a value.

On the tab ***Value display***, you can configure the display for the values "Defined" and "Undefined". You can define whether the date and/or time should be displayed for the element. To do so, activate the corresponding options ("Show date" and/or "Show time").

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes.

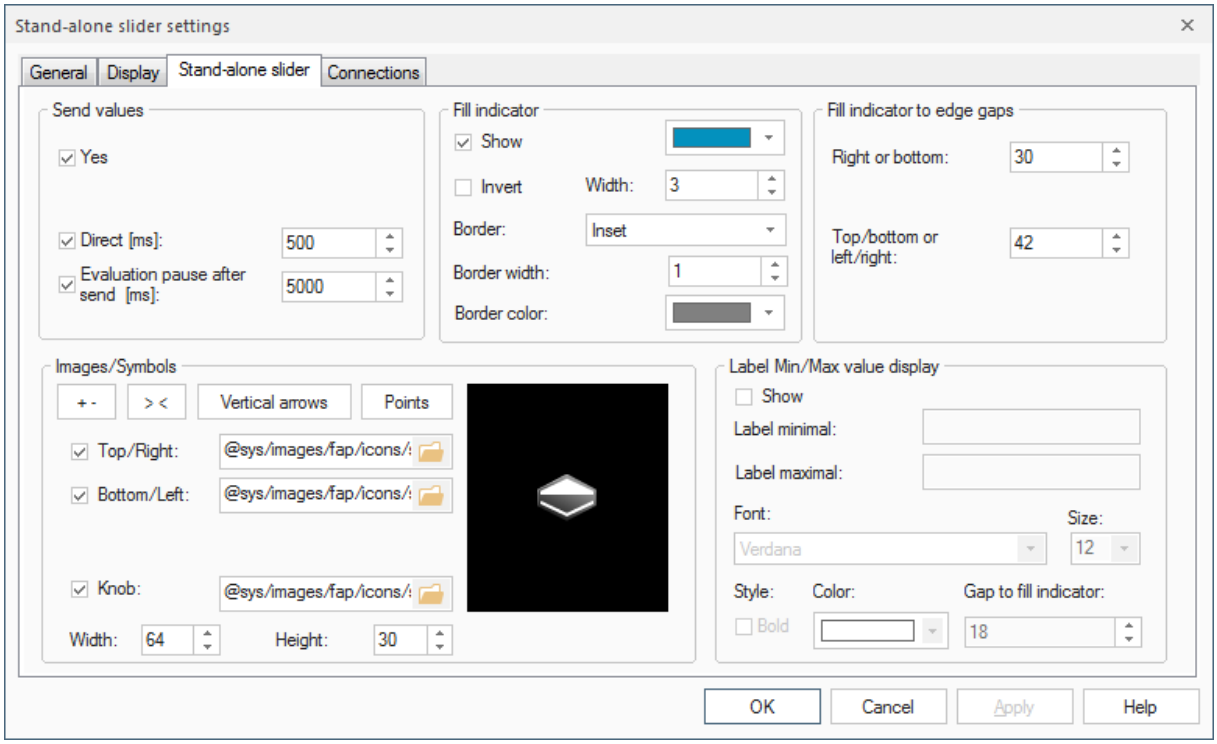
5.2.2.23 Stand-Alone Slider

The slider exists in two versions, one in horizontal and in a vertical representation.

On the tab ***General*** you can define a name as well as the limits for the element.

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes or the function ***Send value*** to send entered values.

The tab ***Stand-alone slider settings***:



Fill indicator

The fill indicator will only left (at a horizontal slider) or at the top (at a vertical slider) be displayed.

Show: Activates/deactivates the display of the fill indicator.
Invert: Inverts the minimum and maximum display.
Color: Color of the fill indicator.
Width: Width of the fill indicator (in pixels).

Border settings: Optional settings for the border of the fill indicator (Set *Border at* "None" to deactivate the border).

Fill indicator to edge gaps

The gaps between the fill indicator and the edge of the display elements.

Send values

Yes: Activates/deactivates the sending of values.
Direct [ms]: Direct transmission of the value.
Evaluation pause after send [ms]: After the user finished the regulation procedure, an evaluation pause occurs which is to be set here (in ms).

Images/Symbols

Detailed settings for the display of the slider can be done here. It exists a selection of standard sliders. With a click at **+-** , **> <**, **Vertical arrows** or **Points**, the appropriate image settings will be loaded.
In case that the slider should be even more individually configured, separate images can be loaded as well.
Width and **Height** only refer to the image of the knob.

Label Min/Max value

Show: Activates/deactivates the display of the fill indicator display mark.
Label minimal: Display for the minimal value (e.g. "dark").
Label maximal: Display for the maximal value (e.g. "bright").
Font, Size, Color, Style set the presentation of the fill indicator display mark.
Gap to fill indicator: Defines the gap between the mark and the fill indicator itself.

5.2.2.24 Slider with Display

The stand-alone sliders with display combine a horizontal slider with the text / value / icon elements. There is a stand-alone slider with display for temperature, blind or dimmer.

Settings for the tabs can be found in the notes on general settings that can be made in the property dialogs of the display elements:

Settings	*Chapter*	
General	5.2.1.1	Tab General
Display	5.2.1.3	Tab Display
Value display	5.2.1.4	Tab Value Display
Stand-alone slider	5.2.2.23	Stand-alone Slider

On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes or the function ***Send value*** to send entered values.

The operation within the visualization server depends on the selection of the slider with display. These differ as follows:

Slider with display for temperature:

Value range: Variably adjustable on the tab *General*.
Send behavior in the server:
Short keystroke: Value change in 0.5° steps.
Long keystroke: Jump to minimum or maximum value.

Slider with display for blind:

Value range: Value range between 0 and 100.
Send behavior in the server:
Short keystroke: Up/down command for blind (requires connection to a binary data point).
ATTENTION: The element does only require the short run command!
Long keystroke: Jump to minimum or maximum value.

Slider with display for dimmer:

Value range: Value range between 0 and 100.
Send behavior in the server:
Short keystroke: Switching the light on and off (requires connection to a binary data point).
ATTENTION: The element does not require a callback variable!
Long keystroke: Jump to minimum or maximum value.

5.2.2.25 Double Stand-Alone Slider with Blind / Slat Display

The double stand-alone slider combines the control for the blind and slat.

Settings for the tabs can be found in the notes on general settings that can be made in the property dialogs of the display elements:

Settings	*Chapter*	
General	5.2.1.1.1	Element Name
	5.2.1.1.2	Label
Display	5.2.1.3	Tab Display
Stand-alone Slider 1	5.2.2.23	Stand-alone Slider
Stand-alone Slider 2	5.2.2.23	Stand-alone Slider
Value display 1	5.2.1.4	Tab Value Display
Value display 2	5.2.1.4	Tab Value Display

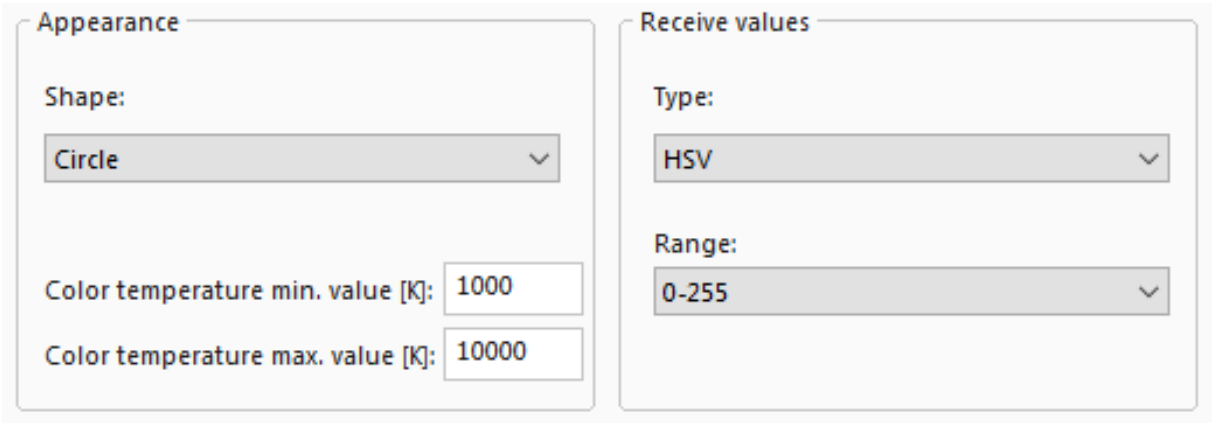
On the tab ***Connections***, assign the function ***Only update*** to the connected process variable to display incoming value changes or the function ***Send value*** to send entered values.

Send behavior in the server:

Short keystroke: Up/down command for blind / step for slat (requires connection to a binary data point).
ATTENTION: The element does only require the short run command!
Long keystroke: Jump to minimum or maximum value.

5.2.2.26 Color Input

TAB *COLOR INPUT*



Appearance

Shape: Specifies whether the input color is shown as a circle or a rectangle

Color temperature min/max: Sets the range for the color temperature.
Range: 0-65535

Receive values

Type: Sets the value type which is received and interpreted.
Selection: RGB, HSV, Color temperature

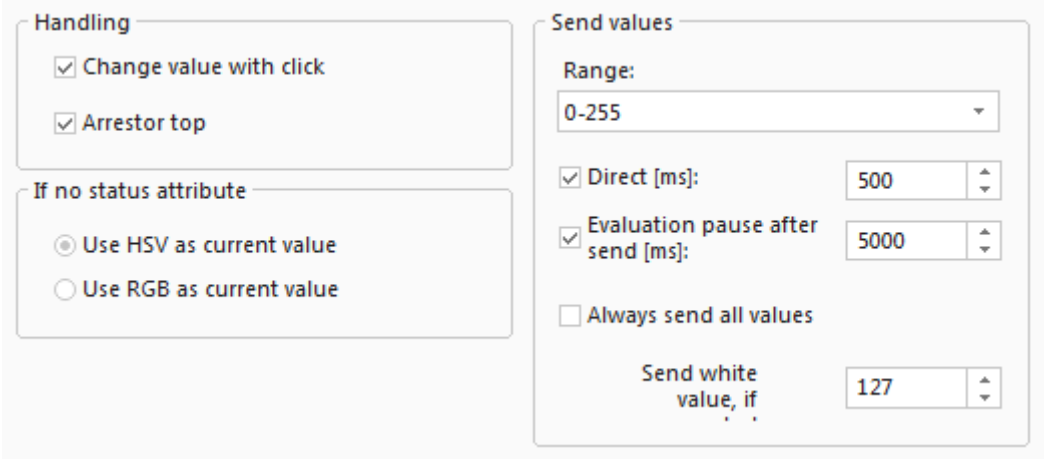
Range: Defines the value range.
Selection: 0-100 and 0-255

Type	**Selection**
RGB	0-100, 0-255
HSV	0-100, 0-255
Color temperature	0-100, 0-255, 0-65535
	Color value in Kelvin, brightness 0-100
	Color value in Kelvin, brightness 0-255
	Color value in Kelvin, brightness 0-65535

Color value in Kelvin means that the control receives values in the range between the defined minimum and maximum.

5.2.2.27 Colour Control

TAB *COLOUR CONTROL*



Handling	**Change value with click:** If this function is *activated,* the colour wheel can be changed with a click at the desired point. Is it *deactivated,* the colour wheel can only be changed by holding and turning of the slider area. **Arrestor top:** Is the function *activated,* the knob cannot be turned over the upper circle middle (where minimum and maximum meet).
If no status attribute	If multiple data points are connected that send different units (HSV, RGB), it can be set here which unit should be shown at arrival.
Send values	**Direct [ms]:** Direct transmission of the value. **Evaluation pause after send [ms]:** After the user finished the regulation procedure, an evaluation pause occurs which is to be set here (in ms). **Always send all values:** Sends all values. **Send white value, if connected:** Specifies with white value (0-255) should be sent.

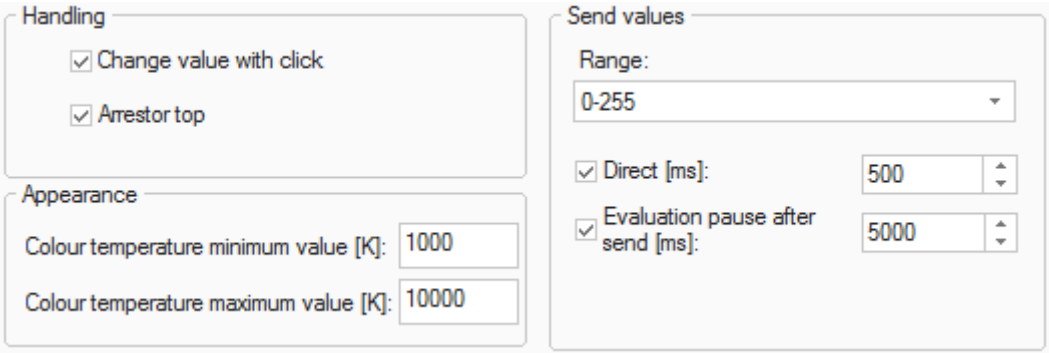
Important: The Colour Control will always send values between 0-100 or 0-255. Make sure that you do not use units like 5.001 DPT_Scaling for group addresses in the KNX Editor, which you are planning to use with a Colour Control, because these units also convert the value and thus distort the result.

5.2.2.28 Tunable White Control

On the tab ***General*** you can define a name as well as the limits for the element.

A difference between this and other elements is for example the entry of the ***Minimum*** and ***Maximum value***. Here, the standard is from 1 000 to 10 000, because the unit is ***Kelvin***.

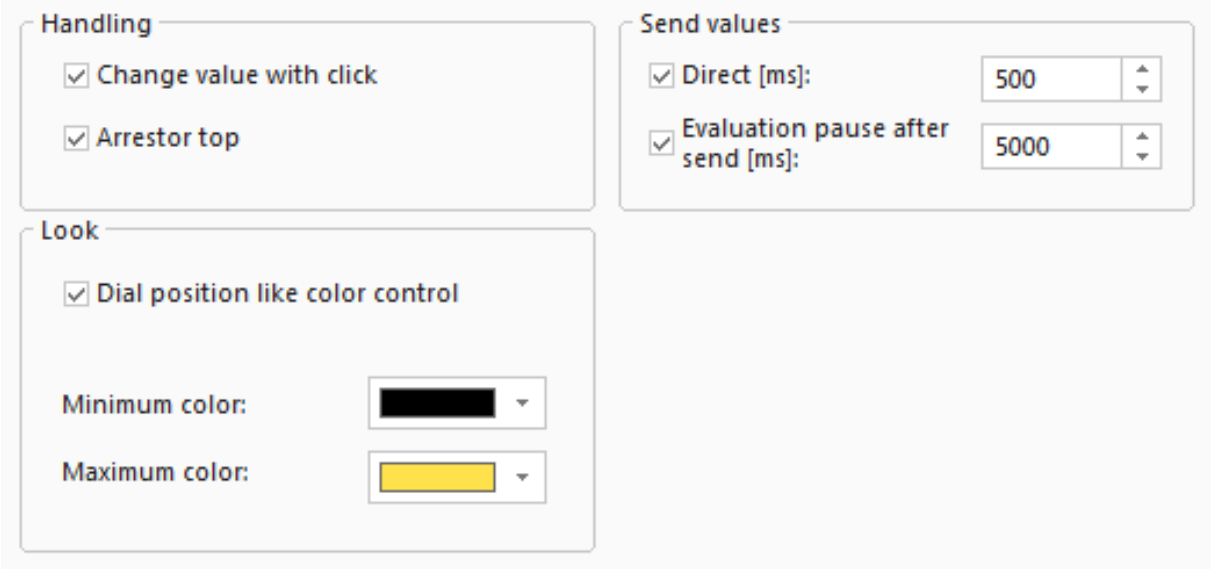
TAB *TUNABLE WHITE CONTROL*



Handling	**Change value with click:** If this function is *activated,* the colour wheel can be changed with a click at the desired point. Is it *deactivated,* the colour wheel can only be changed by holding and turning of the slider area. **Arrestor top:** Is the function *activated,* the knob cannot be turned over the upper circle middle (where minimum and maximum meet).
Appearance	**Color temperature min/max:** Sets the range for the color temperature. Range: 0-65535
Send values	**Range:** Defines the value range. Selection: 0-100, 0-255, 0-65535 Color value in Kelvin. Brightness 0-100, Color value in Kelvin. Brightness 0-255, Color value in Kelvin. Brightness 0-65535 Color value in Kelvin means that the control sends values in the range between the defined minimum and maximum. **Direct [ms]:** Direct transmission of the value. **Evaluation pause after send [ms]:** After the user finished the regulation procedure, an evaluation pause occurs which is to be set here (in ms).

5.2.2.29 Knob

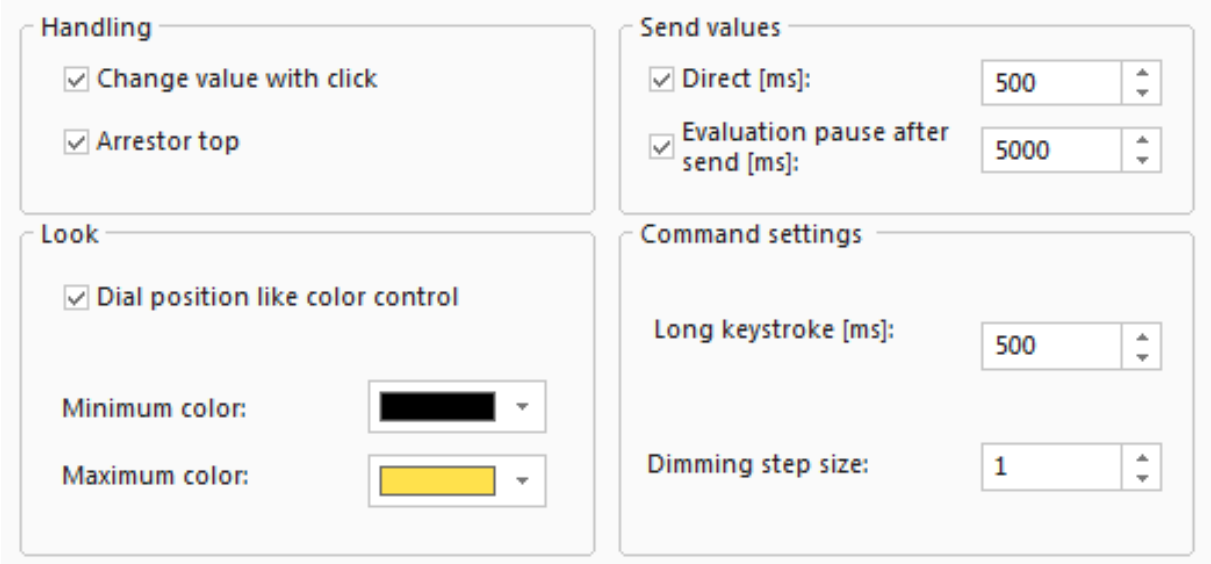
TAB *KNOB*



Handling	**Change value with click:** If this function is *activated,* the colour wheel can be changed with a click at the desired point. Is it *deactivated,* the colour wheel can only be changed by holding and turning of the slider area. **Arrestor top:** Is the function *activated,* the knob cannot be turned over the upper circle middle (where minimum and maximum meet).
Look	**Dial position like color control:** Cosmetic change of the layout to look similar to the colour control element. If deactivated, the circle within the element will be displayed in the middle and bigger than before.
Send values	**Direct [ms]:** Direct transmission of the value. **Evaluation pause after send [ms]:** After the user finished the regulation procedure, an evaluation pause occurs which is to be set here (in ms).

5.2.2.30 Dimmer Knob

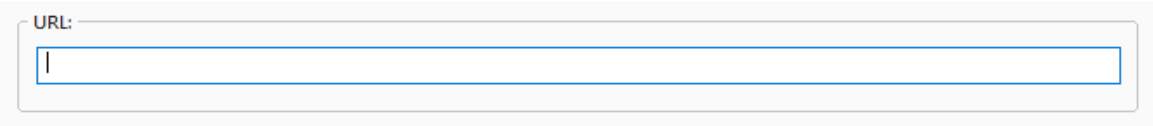
TAB *KNOB*



Handling	**Change value with click:** If this function is *activated,* the colour wheel can be changed with a click at the desired point. Is it *deactivated,* the colour wheel can only be changed by holding and turning of the slider area. **Arrestor top:** Is the function *activated,* the knob cannot be turned over the upper circle middle (where minimum and maximum meet).
Look	**Dial position like color control:** Cosmetic change of the layout to look similar to the colour control element. If deactivated, the circle within the element will be displayed in the middle and bigger than before.
Send values	**Direct [ms]:** Direct transmission of the value. **Evaluation pause after send [ms]:** After the user finished the regulation procedure, an evaluation pause occurs which is to be set here (in ms).
Command settings	In contrast to the usual knob, special settings similar to the dimmer display element can be done here.

5.2.2.31 Embedded Site

The ***embedded site*** is an element with which it is possible to display content of external HTML 5 pages.



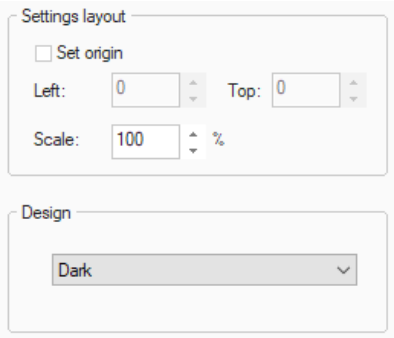
NOTE: The address of the external page must have the prefix **http://** to make sure, that the address can be resolved correctly.
Furthermore, there are several pages that can not be embedded (for example *http://google.com*). This is a special setting that is sent from the page itself (*X-Frame-Options: Deny* in the header).

Hint: With the button and the button action **toggle embedded site** the content of the control can be changed.

5.2.3 16 Channel Weekly Time Switch

5.2.3.1 Control Settings

TAB *OVERLAY LAYOUT*

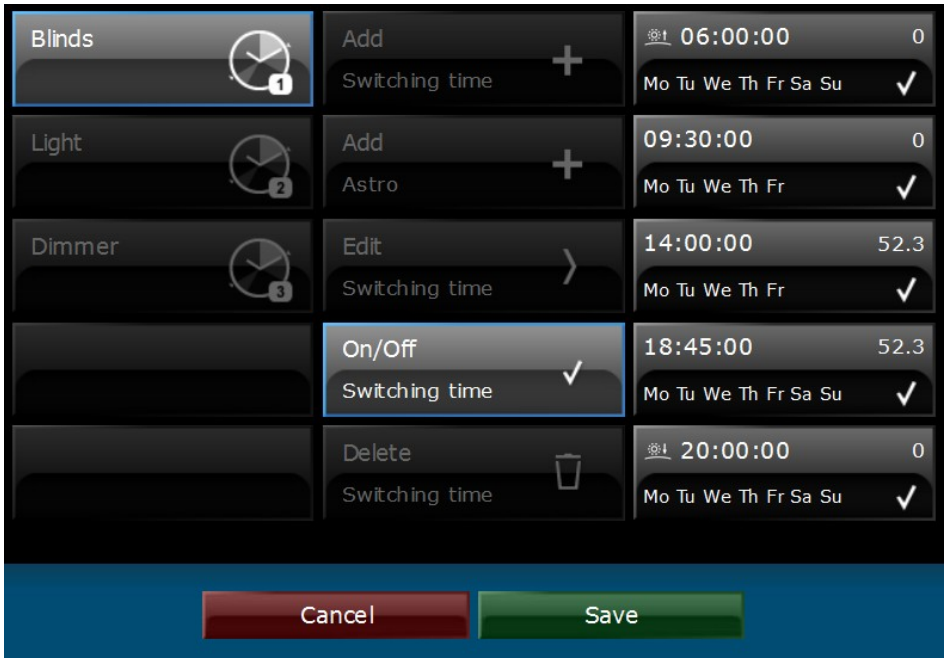


In the dialog ***Settings layout*** you can define the positioning of the overlay for the settings of the weekly time switch.

The ***Design*** option determines the color palette of the overlay, which is displayed in the server after pressing the button.

5.2.3.2 Operation via HTML User Interface

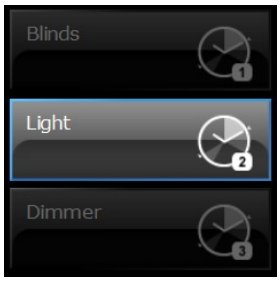
NOTE: Please note that the ***overlay for the settings of the 16 channel weekly time switch*** can **ONLY** be displayed in the HTML user interface.



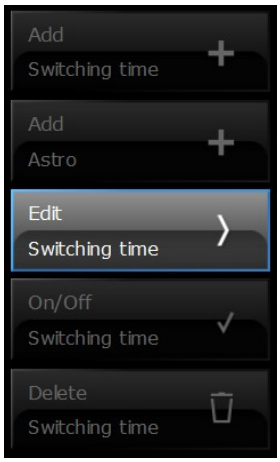
The weekly time switch has two display modes: **Landscape** is the version with three columns. When space is limited, the weekly time switch is displayed in the single-column **portrait** mode.
There are 4 columns:
Column 1: Channel list
Column 2: Action selection list
Column 3: List of the programmed switching commands
Column 4: Switching command details

Clicking the button ***Back to summary*** in the upper left corner of the screen closes the overlay. All unsaved changes will be discarded.

5.2.3.2.1 Column Contents



Channel list
List of available channels.
Select the according channel for creating and editing switching commands.



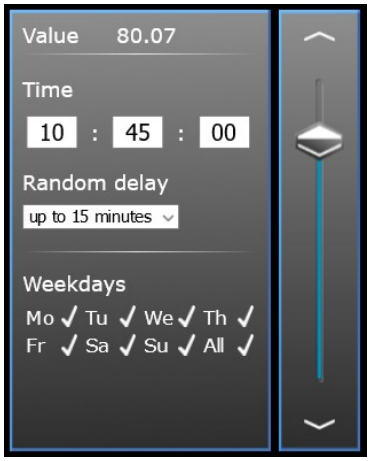
Action selection list
If you have selected a channel, the following options are available: ***Add, Edit, On/Off*** and ***Delete*** of switching commands.

List of the programmed switching commands
Switching commands with time and day format, the value and the status active or inactive (check mark).
The sun icon marks astro clock switching commands. The displayed time is the calculated sunrise or sunset time of today. Since these times may be different for each day of the year, the displayed time is only a guideline.

Switching command details

Here you create the exact configuration. Details on configuration of switching commands are given in the following chapter **5.2.3.2.2**, details on configuration of astro commands in chapter **5.2.3.2.3**.

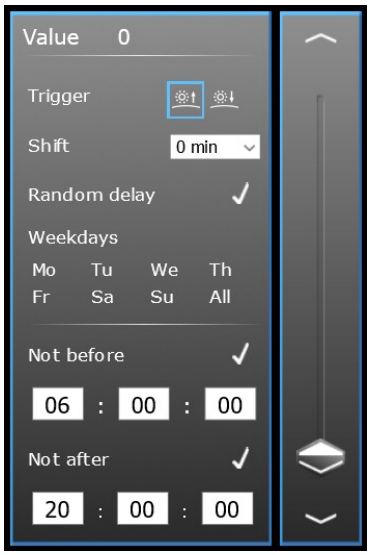
5.2.3.2.2 Create and Configure Switching Commands



The value is set by using the slider.
When setting the clock (hh/mm/ss) consider that only 5 second intervals are accepted.
When triggering time events, a random delay in 15-minute increments up to a maximum of one hour can be selected.
With a set time of *10:45* and a selected random delay of *up to 15 minutes*, the switching command is executed at some time between 10:45 and 11:00.
Activated days are tagged with a check mark, inactive days don't have a symbol.

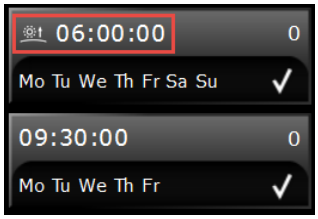
5.2.3.2.3 Create and Configure Astro Commands

Switching times can also be dynamically adjusted according to the position of the sun.



The value is set by using the slider.
The trigger is either the sunrise or the sunset.
Clicking on *Random delay* will activate it for the astro command. Thereby the command will not be executed on sunrise/sunset, but with a random delay of up to 5 minutes.
Additionally, a time difference to sunrise or sunset can be set, selectable in 5-minute steps from -60 to +60 minutes.
Activated days are tagged with a check mark, inactive days don't have a symbol.
If a time is entered for *Not before* or *Not after*, the switching command is not executed before or rather after this entered time. If the sunrise (e.g. 05:30 o'clock) was set as the trigger for a switching command and you entered 6:00:00 as *Not before*, the switching command is only executed at 6 o'clock. Click on the lettering of *Not before* or *Not after* to activate or deactivate the function.
When setting the clock (hh/mm/ss) consider that only 5 second intervals are accepted.

Note: Several sunrise/sunset commands can only be created on the same day if the entries at *Shift* and *Not before* and *Not after*, if activated, differ.

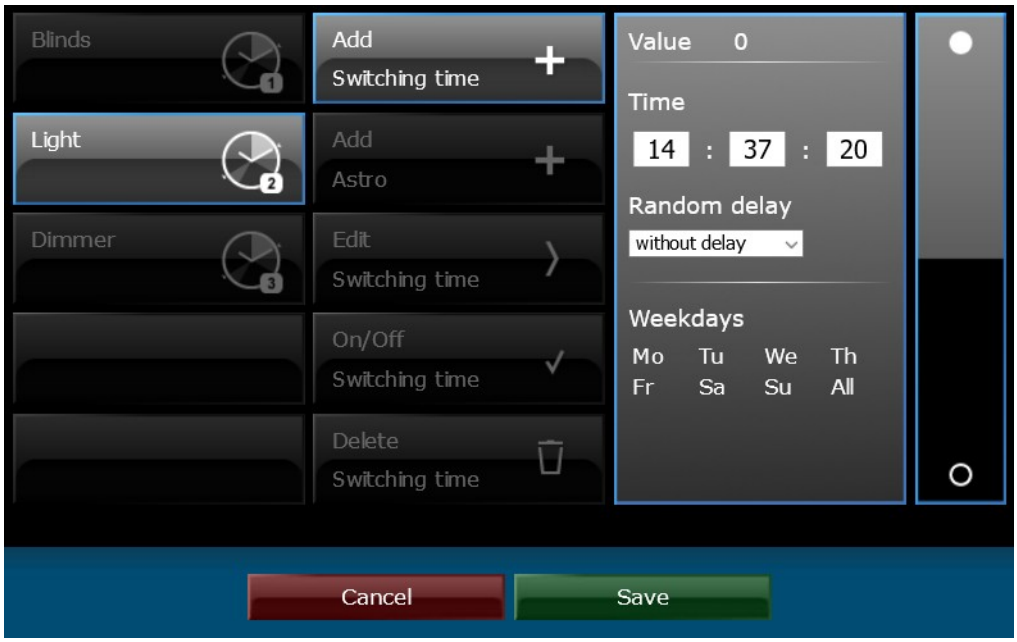


In the list of programmed switching commands, astro clock switching commands are marked with a sun icon. The displayed time is the calculated sunrise or sunset time of today. Since these times may be different for each day of the year, the displayed time is only a guideline.

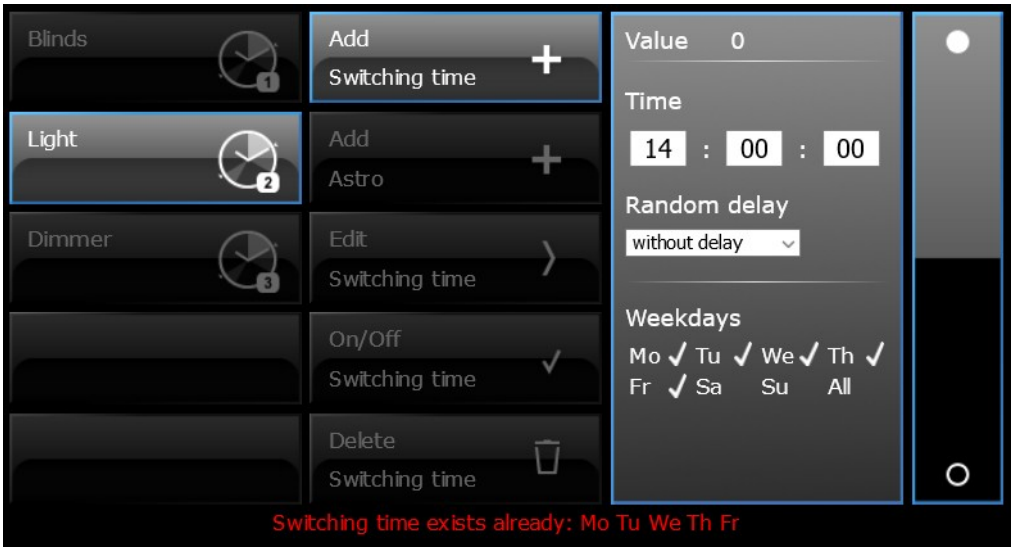
5.2.3.2.4 Creating a Switching Command

Add switching time

Step 1: Select the required channel.
Step 2: Press the ***Add Switching time*** or ***Add Astro*** button.
Step 3: Make user-specific settings in the switching command details.
Step 4: Click on the ***Save*** button, a short note in the dialogue of switching command details will confirm the successfully saving.



If there exists already a switching command for a selected day at a specified time, the following message is displayed



Adjust the switching time and save the change again.

Note: You can create up to a maximum of 16 switching times per channel!

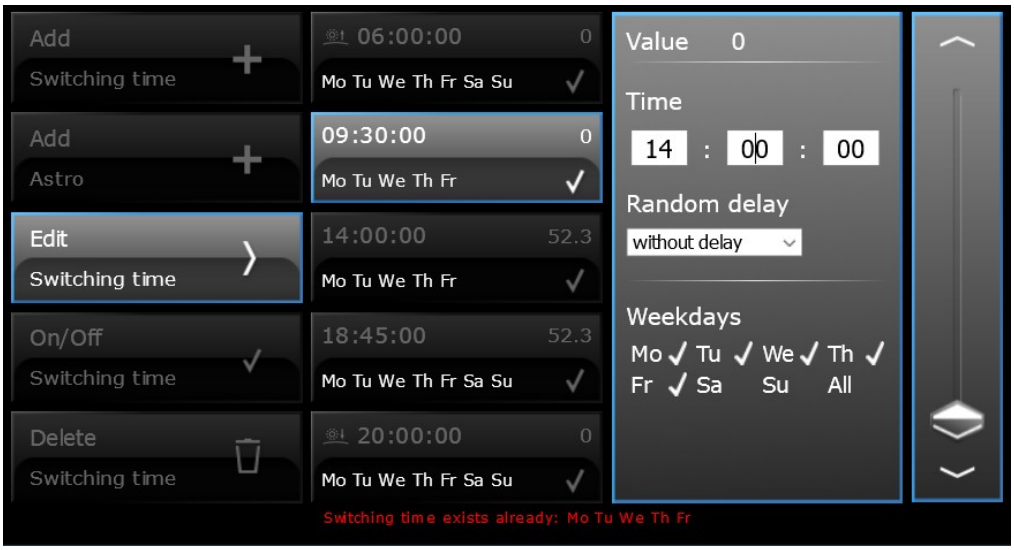
Note: With analogue variables, the adjustable values range between 0 and 100, unless other values are defined in the process model in the variables under Min. and Max. With binary variables "on" and "off" are preset.

Note: Pressing the "Cancel" or "Back to summary" button, reloading the site again (F5) or selecting another element, discards all changes.

Edit switching time

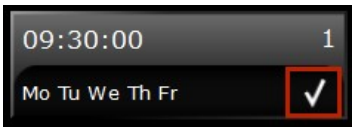
Step 1: Select the required channel.
Step 2: Press the ***Edit*** button.
Step 3: Select the required switching command in the list of the programmed switching commands.
Step 4: Make the desired changes in the switching command details.
Step 5: Click on the ***Save*** button, a short note in the dialogue of switching command details will confirm the successfully saving.

If there exists already a switching command for a selected day at a specified time, the corresponding message is also displayed here:



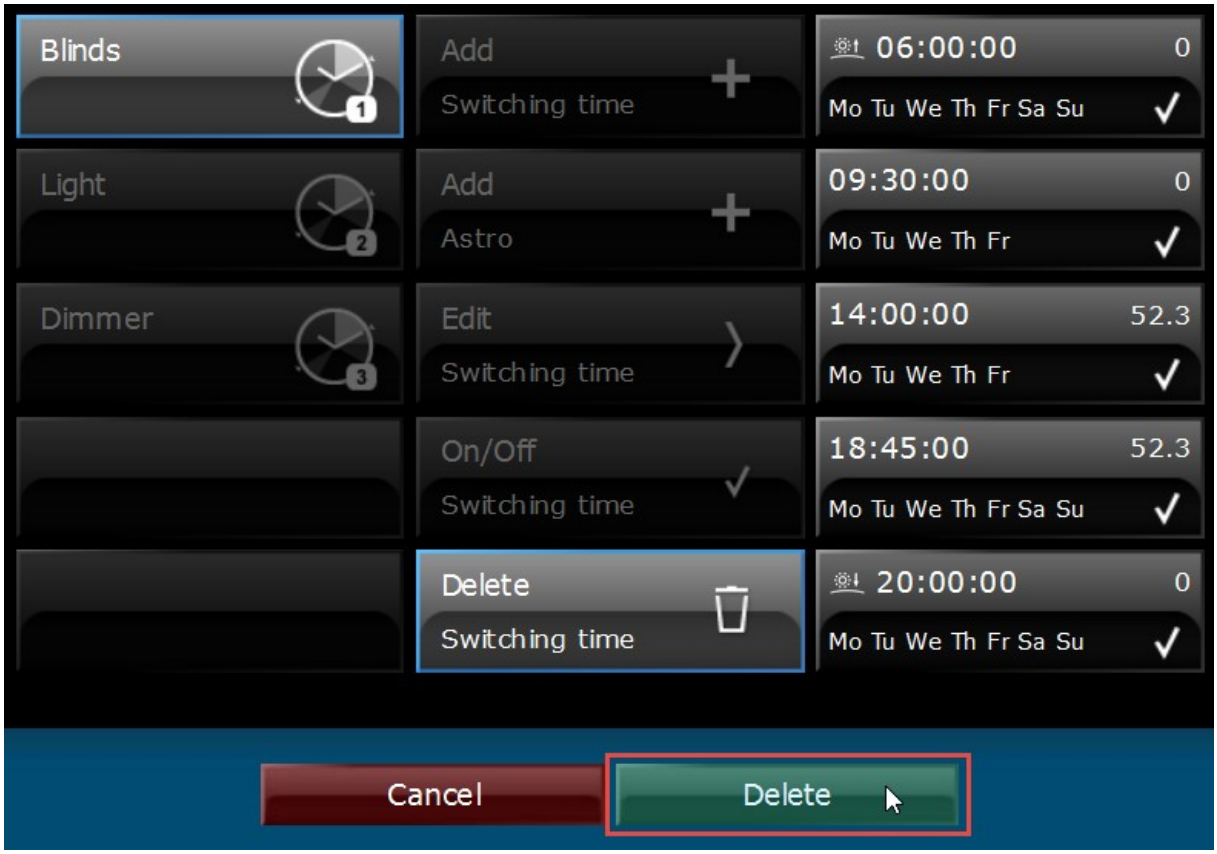
Switching time On/Off

Step 1: Select the required channel.
Step 2: Press the ***On/Off*** button.
Step 3: Clicking the required switching command in the switching command list activates
or deactivates the switching command (check mark).
Step 4: Clicking the ***Save*** button accepts the changes without note.
Clicking the ***Cancel*** button restores all original settings.



Delete switching times

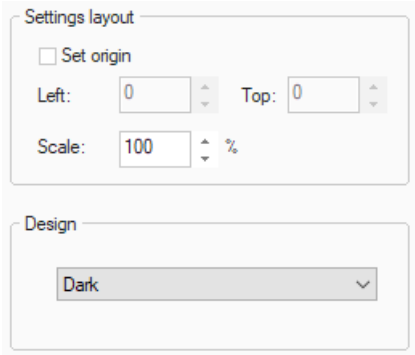
Step 1: Select the required channel.
Step 2: Press the ***Delete*** button.
Step 3: Select the switching time to delete
Step 4: Clicking the ***Delete*** button immediately deletes the selected switching time.
Clicking the ***Cancel*** button switches to the edit mode.



5.2.4 16 Channel Store Scene with ACK

5.2.4.1 Control Settings

TAB *OVERLAY LAYOUT*



With the layout settings, you can set the position of the overlay to display the memory scene on the screen.

The ***Design*** option determines the color palette of the overlay, which is displayed in the server after pressing the button.

5.2.4.2 Operation via HTML User Interface

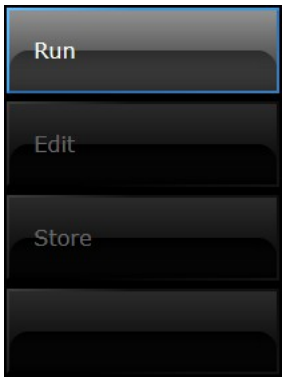
NOTE: Please note that the ***overlay for the settings of the 16 channel store scene with ACK*** can **ONLY** be displayed in the HTML user interface.

The store scene with ACK has two display modes: **Landscape** is the version with three columns. When space is limited, the store scene is displayed in the single column **portrait** mode.

3 columns are distinguished:
Column 1: Action selection list
Column 2: Channel list
Column 3: Channel details

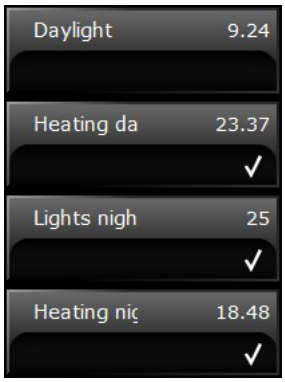
Clicking the button ***Back to summary*** in the upper left corner of the screen closes the overlay. All unsaved changes will be discarded.

5.2.4.2.1 Column Contents

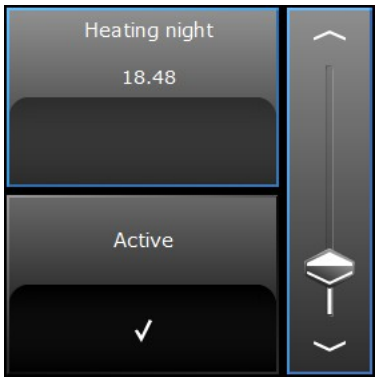


Action list

The following options are available:
Run, edit, store.



Channel list
Displays the channels of the scene.



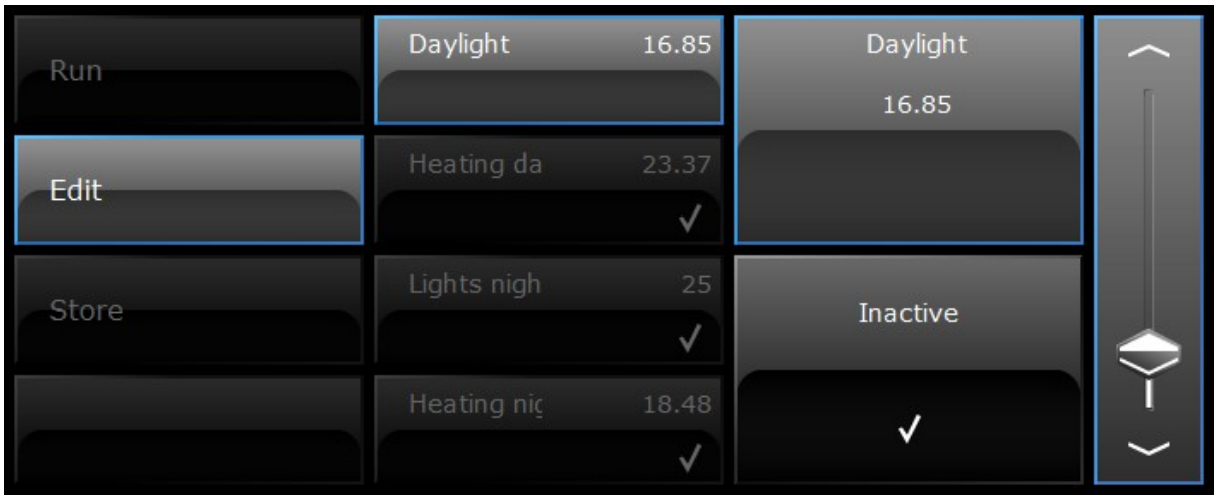
Channel details
The value is set by the slider. Furthermore, the channel can be enabled or disabled here.

Note: The displayed values are the current values in the feedback object, not from the scene memory.

5.2.4.2.2 Using of the Overlay

Edit channel

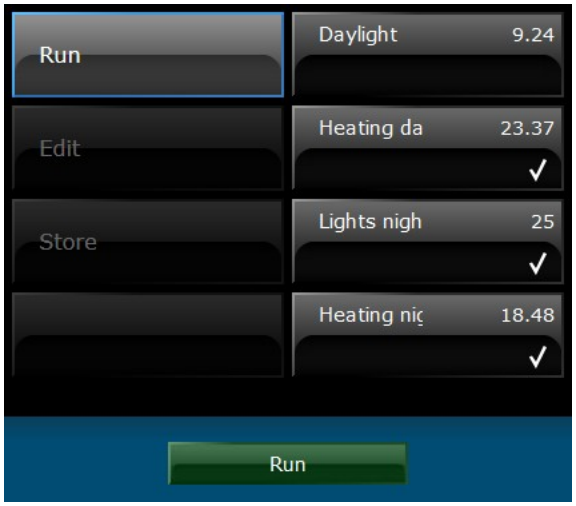
Step 1: Select ***edit***.
Step 2: Choose channel.
Step 3: Change the settings in the channel details.
Step 4: Select ***store***.
Step 5: Click ***save*** to store the scene.



Note: The adjustable values in analog variables are preset from 0 to 100, unless other values are defined in the process model within the variables, under min. and max. value. For binary variables "on" and "off" are preset.

Note: Any unsaved changes will be discarded if not saved after changing or "Back" has been pressed, as well as by reload (F5) of the page or by selecting a different element.

Run scene

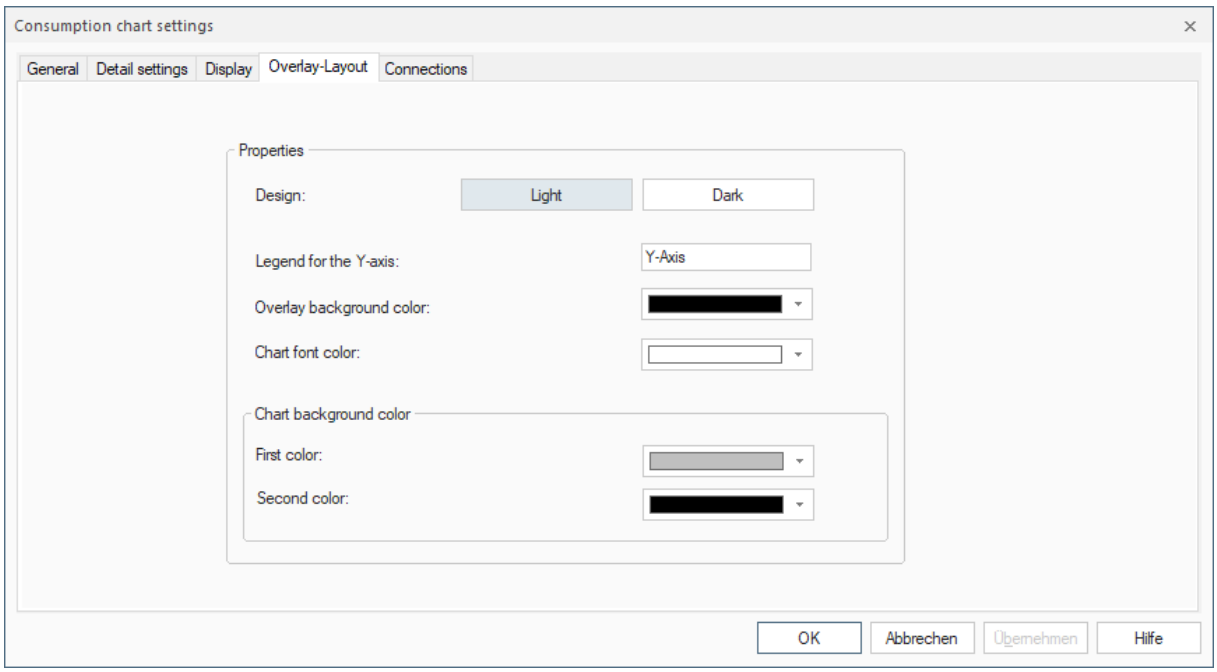


Step 1: Select ***run***.
Step 2: The scene is triggered by pressing the button ***run.***

5.2.5 Charts

5.2.5.1 Configuring the Display Element within the Visualization Editor

TAB *OVERLAY-LAYOUT*



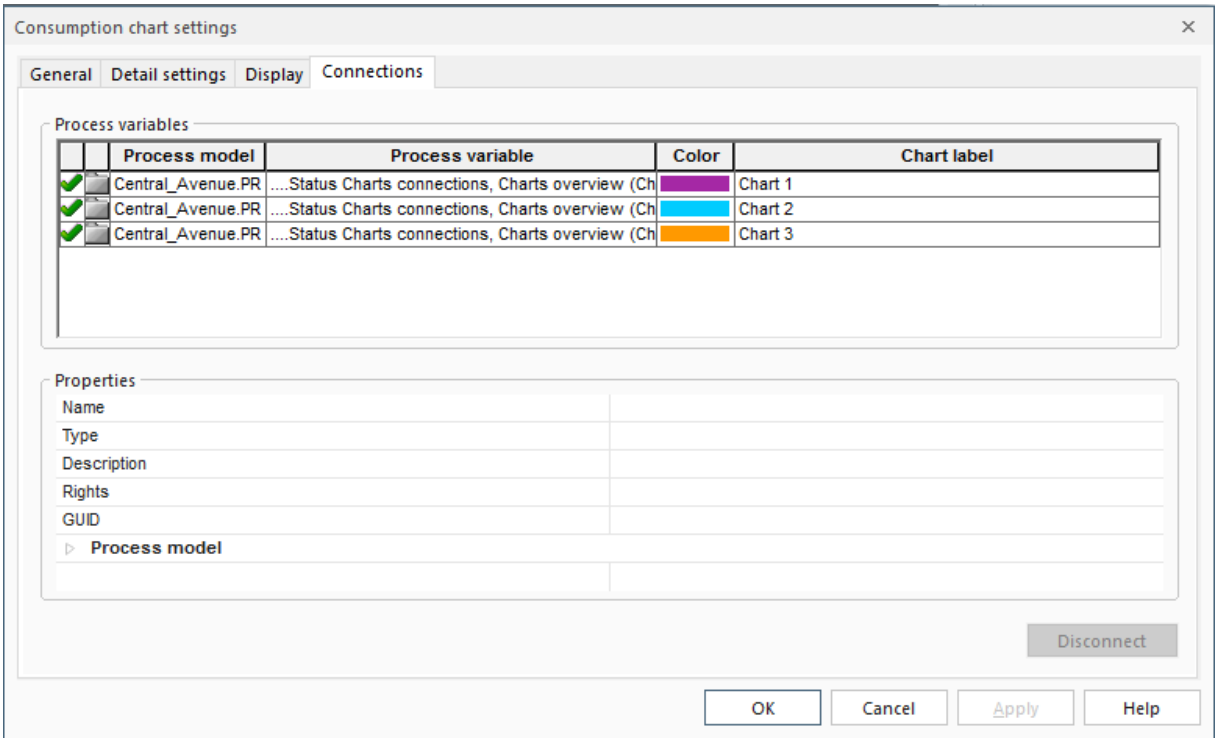
On the tab ***Overlay-Layout*** you can customize the layout of the user interface in the visualization server. The following settings can be made:

Design:	The *Light* and *Dark* options will set all colors to the default values for a bright respectively dark design.
Legend for the y-axis	A label for the y-axis of the graph, displayed to the left of the graph.
Overlay background color	The general background color of the graph page in the visualization server.
Chart font color	The general font color of the graph page in the visualization server.
Chart background colors	These two colors make the color gradient of the graph background.

TAB *CONNECTIONS*

On the tab ***Connections*** special settings for the display of the connected data point curves within the chart can be made.

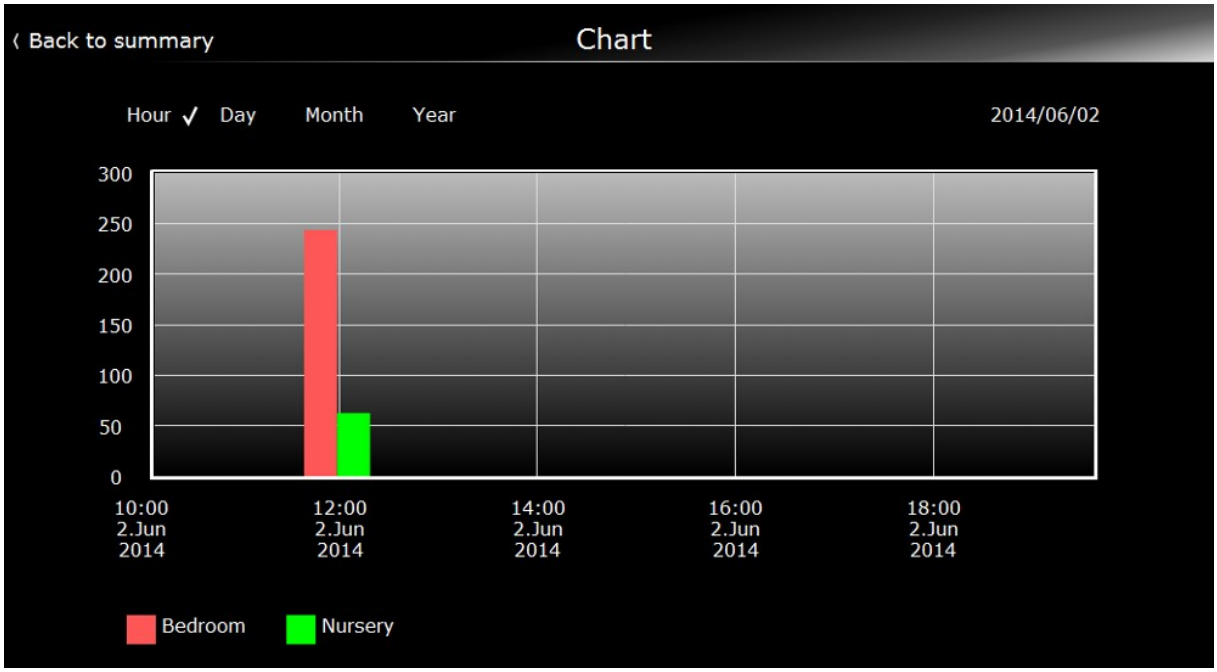
To add a connection, open the connected process in the catalog. Select the desired data point and drag the folder of the consumption data archive on the display element.



Color	The curve of the data point will be displayed in the chart with this color.
Chart label	The name (label) of the data point curve in the chart.

Note: A maximum of 4 data points can be connected with a chart!

5.2.5.2 Handling the HTML User Interface

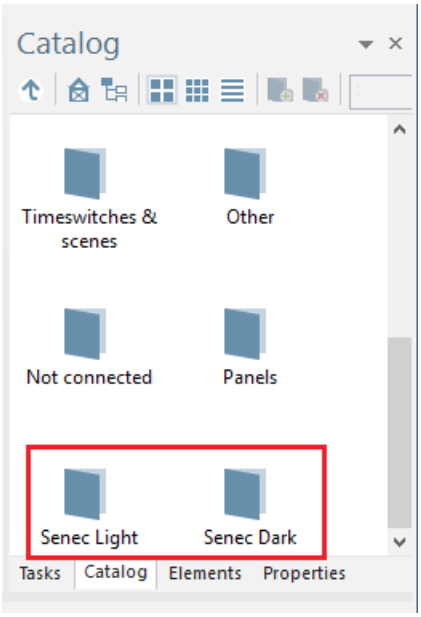


To handle the bar display hold and pull/push the area on the screen.

By clicking on a connected data point in the **legend**, the point can be shown or hidden. **Minute**, **Hour**, **Day**, **Week**, **Month**, **Year** specify the time intervals of the displayed diagram and will define the x-axis. The **x-axis** accurately specifies the time with date (and optionally with time of day). The **y-axis** shows the saved values of the connected variables.

5.3 SENEC-Specific Controls

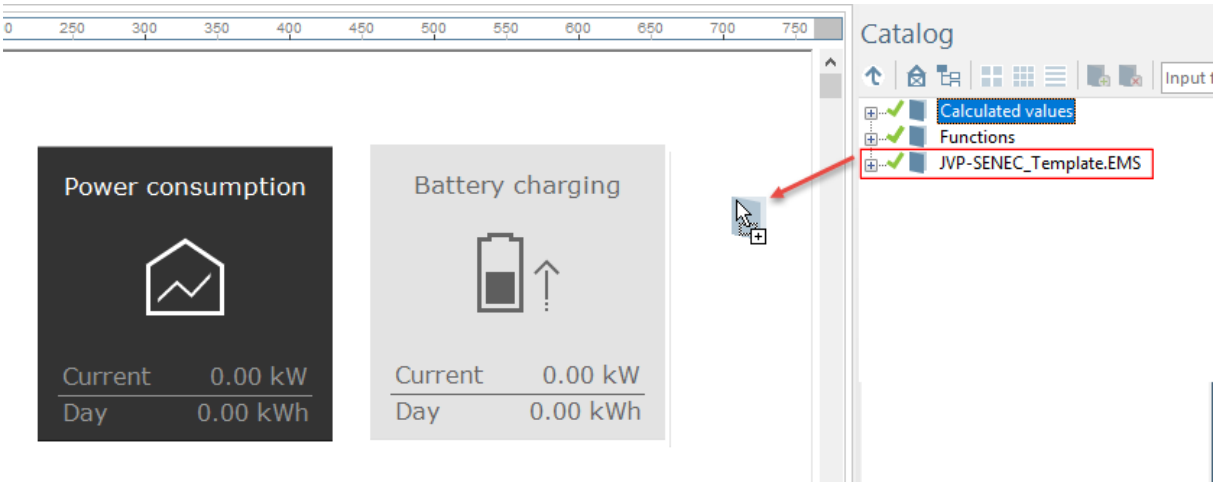
5.3.1 Connect SENEC Controls to Process Variables



In the catalog -> Elements you will find SENEC controls in a light and a dark display variant, which can be connected with an energy management system (short: EMS) project. All elements serve to graphically visualize the various values of a SENEC energy storage device.

Each element that displays energy values has an additional chart overlay that can be opened in the visualization server by clicking on the element.

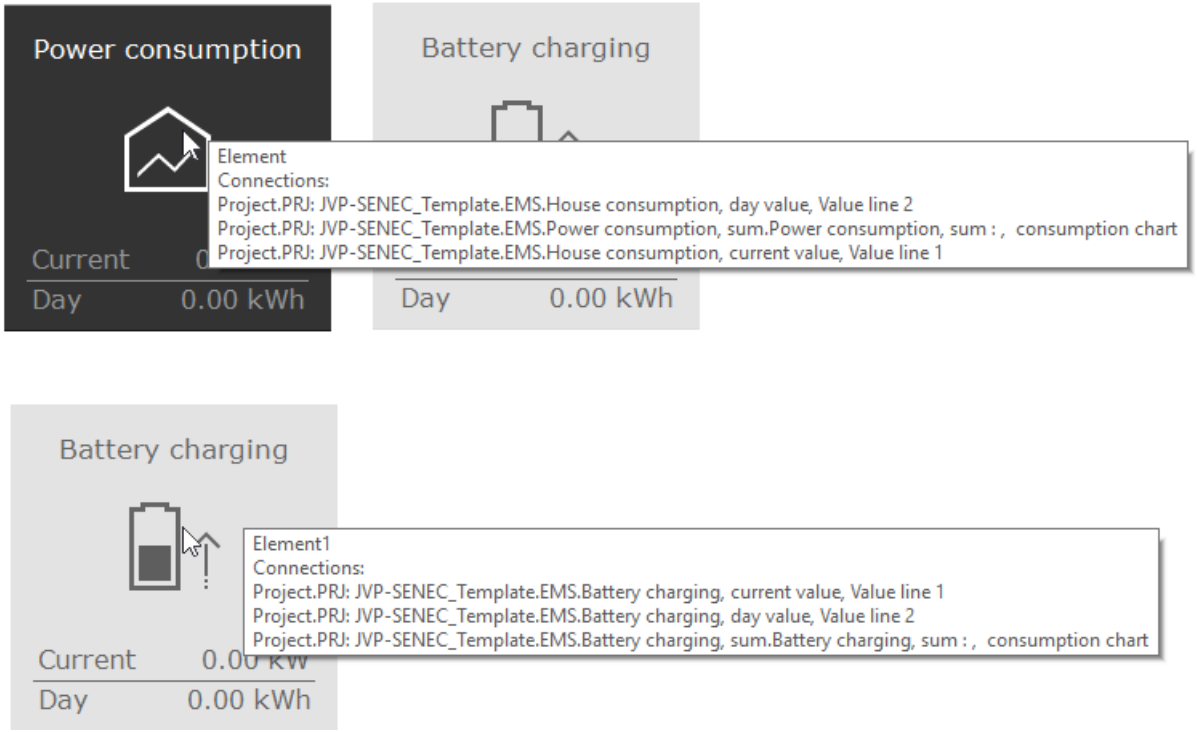
If your worksheet contains one or more SENEC controls, you can easily connect them to EMS data points. To do this, click on Catalog -> Double-click on Process Data -> Double-click on your integrated EMS project. Now drag and drop the project folder with the file extension .EMS onto your **worksheet** and all elements placed there will be connected to the corresponding data points at the same time.



Exception:

- To connect the element ***SENEC control*** to data points, drag the folder ***Functions*** onto the worksheet.
- The function ***Own consumption optimisation*** must have been previously created in the process model and embedded in the visualization editor project.

Move the mouse cursor over the SENEC elements; the newly created connections are displayed:



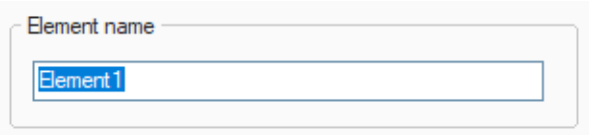
5.3.2 General Settings

The following section provides information on general settings which can be configured in
the property dialog of every control.

5.3.2.1 Tab *General*

5.3.2.1.1 Element Name

The element name is an important specification which facilitates the editing of projects. It serves the internal processing of visualization projects and is not displayed in the visualization server.



The tab ***Elements*** in the tool window ***Elements*** shows all inserted controls under their element name. It is important to assign appropriate element names to the display elements in order to have a better overview over all display elements contained in a worksheet and to facilitate the editing of the worksheets.

An element name **must** be assigned!

You can edit the element name in the property dialog of the display element or in the tool window ***Elements*** on the tab ***Elements***.

An element name may occur only once per worksheet.

Rules for the assignment of element names:

- The following characters are allowed:

  [A-a] to [Z-z]
  [0-9]
  [_] underscore

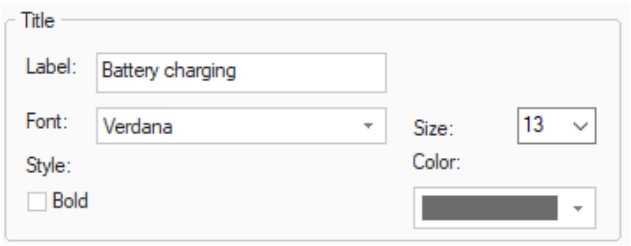
- The following words are **not** allowed:
  and, end, in, repeat, while, break, false, local, return, do, for, nil, then, else, function, not, true, elseif, if, or, until

- The element name **must not** start with a digit

- Element names are **case sensitive**

5.3.2.1.2 Label

Every display element can be labeled. The label is shown on the display element if the layout is configured accordingly.



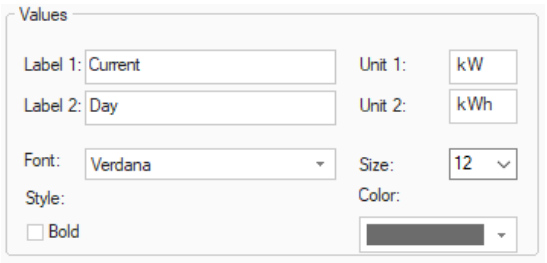
Label: Enter the label.

Font: Select the desired font and font size.

Style: Select the option ***Bold*** to display the label in bold style. *Default: Disabled*.

Color: Select a color from the dropdown list.

5.3.2.1.3 Values



Label 1: Enter the text for the first value.

Label 2: Enter the text for the second value.

Unit 1: Enter the unit for the first value.

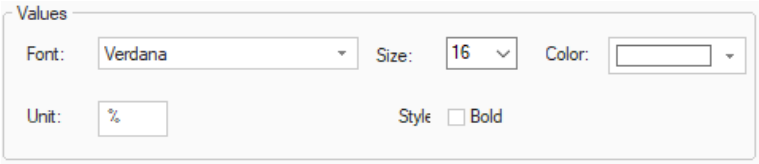
Unit 2: Enter the unit for the second value.

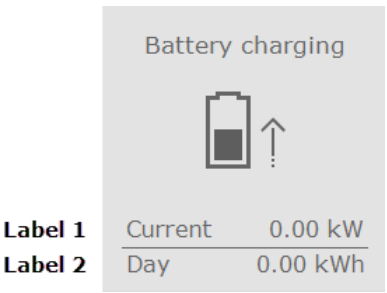
Font: Select the desired font and font size.

Style: Select the option ***Bold*** to display the label in bold style. *Default: Disabled*.

Color: Select a color for Label 1 and Label 2 from the drop-down list.

Deviating display in the dialog of the energy traffic light:





5.3.2.2 Tab *Detail Settings*

The tab detail settings has no effect on the SENEC value elements (current / day values).

Exception: *SENEC control* and *Energy traffic light* are set as described in chapter ***5.2.1.2 Tab Detail Settings***.

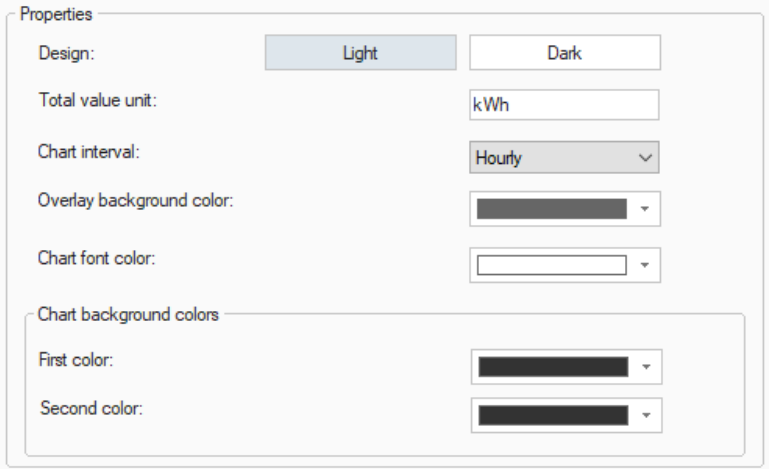
5.3.2.3 Tab *Display*

Applies only to the *Senec Rules* element, settings as described in chapter ***5.2.1.3 Tab Display***.

5.3.2.4 Tab *Overlay-Layout*

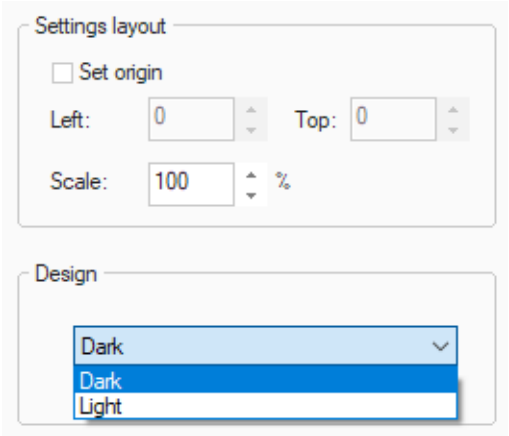
5.3.2.4.1 Properties

On the tab ***Overlay-Layout*** you can customize the layout of the user interface in the visualization server. The following settings can be made:



Design:	The *Light* and *Dark* options will set all colors to the default values for a bright respectively dark design.
Total value unit:	Unit of the total value displayed in the chart.
Chart interval:	Period on the x-axis, possible are *Hourly* and *Monthly*.
Overlay background color:	The background color of the chart page in the visualization server.
Chart font color:	The font color of the chart page in the visualization server.
Chart background colors:	These two colors make the color gradient of the chart background.

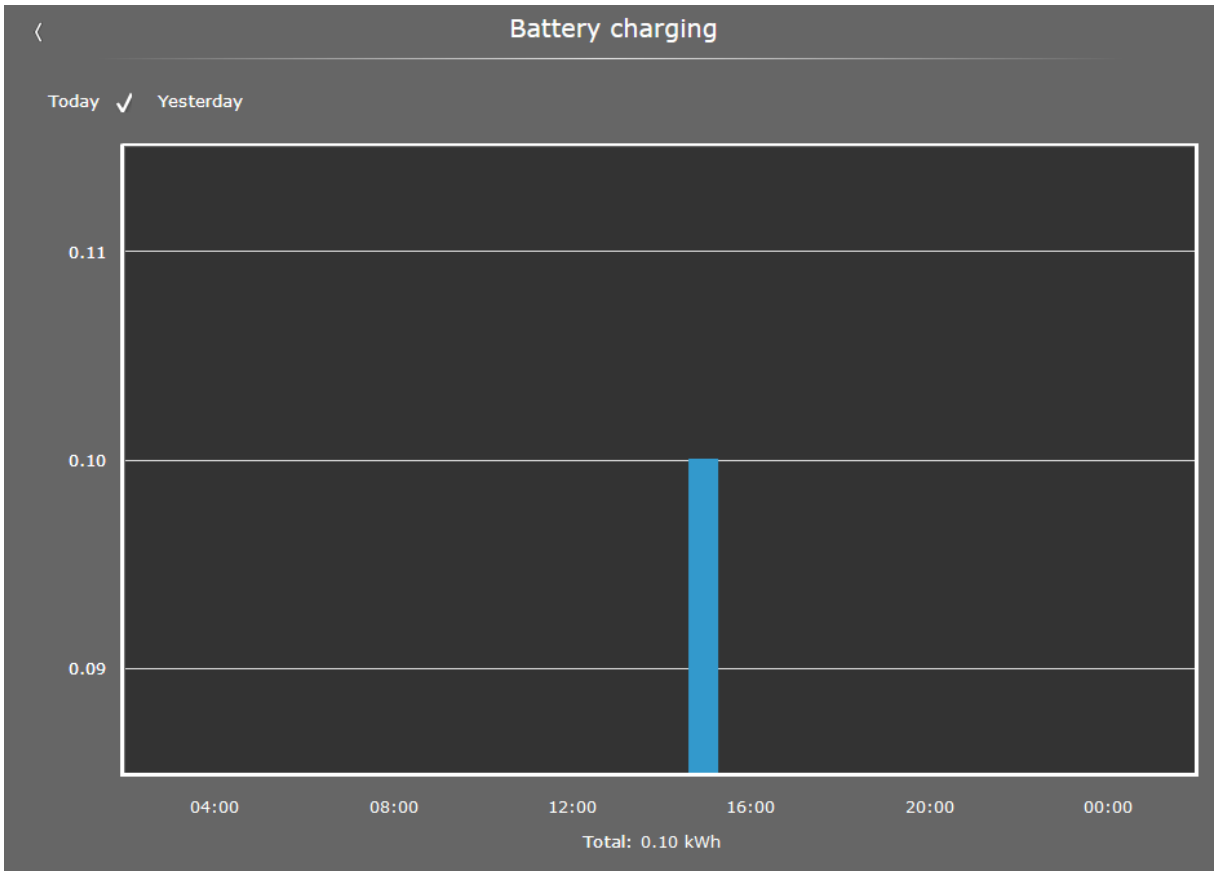
The overlay-layout tab of the ***SENEC rules*** contains the following:



In the dialog ***Settings layout*** you can define the positioning of the overlay.

The ***Design*** option determines the color palette of the overlay, which is displayed in the server after pressing the button.

5.3.2.4.2 Handling the HTML User Interface

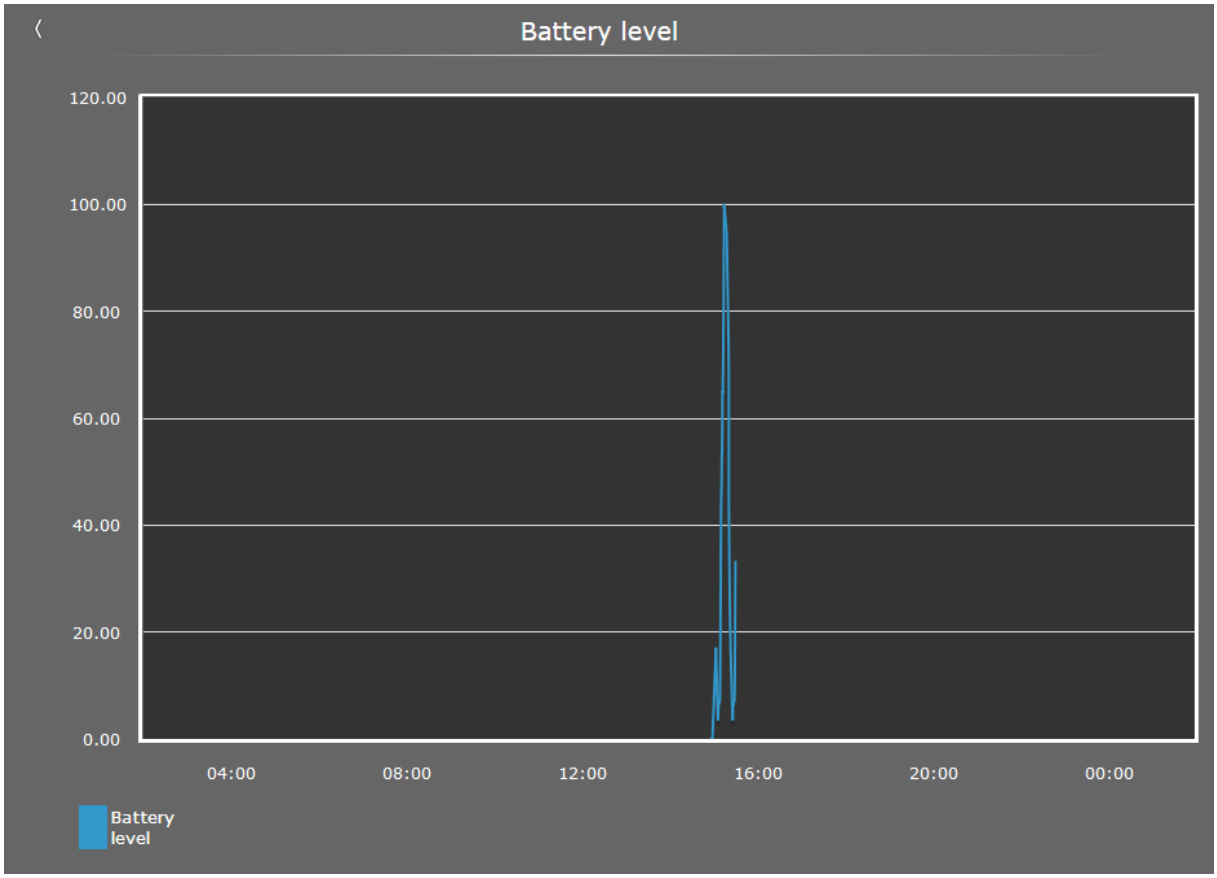


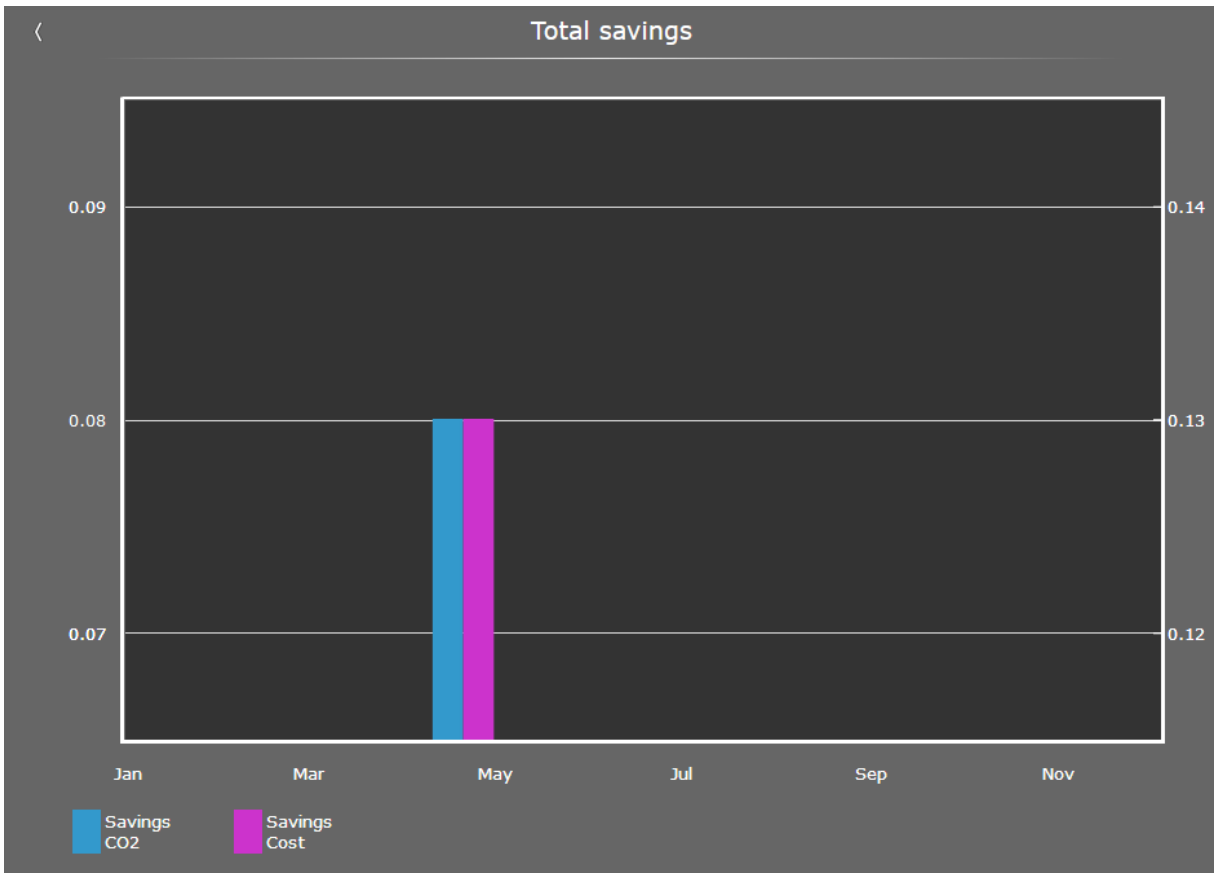
To handle the bar display hold and pull/push the area on the screen.

By clicking on a connected data point in the **legend**, the point can be shown or hidden.

The ***y-axis*** shows the saved values of the connected variables.

The ***x-axis*** specifies the exact period and time of day at SENEC elements with daily values. It is possible to switch between ***Today*** and ***Yesterday***.
At the element *Battery level,* only the current value is displayed.
At the element *Total savings,* the period of one year is displayed by default.


Battery level
120.00
100.00
80.00
60.00
40.00
20.00
0.00
04:00
08:00
12:00
16:00
20:00
00:00
Battery level

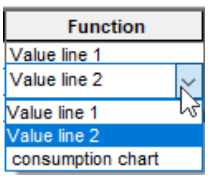

Total savings
0.09
0.08
0.07
0.14
0.13
0.12
Jan
Mar
May
Jul
Sep
Nov
Savings CO2
Savings Cost

5.3.2.5 Tab *Connections*

The tab ***Connections*** shows all process variables connected to the display element. In the column ***Function*** you can assign functions to the process variables.

Click into the column ***Function*** of the desired process variable. From the dropdown list, select the function that you want to assign to the variable.

Function



Value Line 1: Value that is displayed in the first line of the control.

Value Line 2: Value that is displayed in the 2nd line of the control.

Consumption data archive: Archive whose data is displayed in the corresponding chart.

Disconnect:

You can disconnect process variables from the display element: select the desired process variables and click the button ***Disconnect***.

5.3.3 Element Properties

Note: Depending on the system, the values displayed in the visualization may differ from the values displayed in the SENEC portal.

5.3.3.1 Power Consumption

Property	Recommended connections		
Serves to display the power consumption	*Value Line 1:* power consumption, current value	*Value Line 2:* power consumption, day value	*Consumption data archive:* power consumption, sum

5.3.3.2 Battery Charging

Property	Recommended connections		
Serves to display the battery charging	*Value Line 1:* battery charging, current value	*Value Line 2:* battery charging, day value	*Consumption data archive:* battery charging, sum

5.3.3.3 Mains Current Feed-in

Property	Recommended connections		
Serves to display the mains current feed-in	*Value Line 1:* mains current feed-in, current value	*Value Line 2:* mains current feed-in, day value	*Consumption data archive:* mains current feed-in, sum

5.3.3.4 Battery Level

Property	Recommended connections		
Serves to display the battery level	*Value Line 1:* battery level, current value		*Archive:* battery level, events

5.3.3.5 Power Generation

Property	Recommended connections		
Serves to display the power generation	*Value Line 1:* power generation, current value	*Value Line 2:* power generation, day value	*Consumption data archive:* power generation, sum

5.3.3.6 Battery Draw

Property	Recommended connections		
Serves to display the battery draw	*Value Line 1:* battery draw, current value	*Value Line 2:* battery draw, day value	*Consumption data archive:* battery draw, sum

5.3.3.7 Mains Current Draw

Property	Recommended connections		
Serves to display the mains current draw	*Value Line 1:* mains current draw, current value	*Value Line 2:* mains current draw, day value	*Consumption data archive:* mains current draw, sum

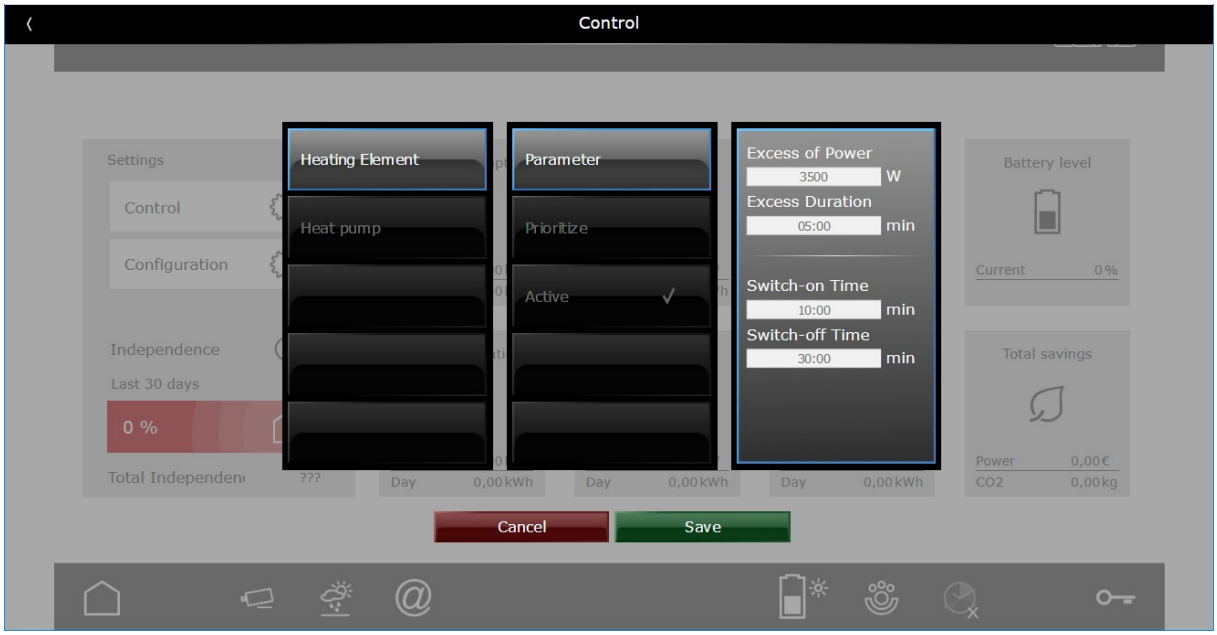
5.3.3.8 Total Savings

Property	Recommended connections		
Savings since the connection of the visualization to the storage was established or since the last reset of the savings values	*Value Line 1:* power cost savings: current value	*Value Line 2:* CO2 savings: current value	*Consumption data archive:* electricity costs savings *Consumption data archive:* CO2 savings

5.3.3.9 SENEC Control

Property	Recommended connections
Serves to display the own consumption optimisation	*Function:* own consumption optimisation

5.3.3.9.1 Handling the HTML User Interface



Energy surplus: Threshold that the energy surplus must at least have.

Duration surplus: Duration, which the surplus must be above the set threshold.

Switch-on time: Minimum switch-on time of load.

Switch-off time: Minimum switch-off time of load.

The operation mode is described in the process model manual in chapter ***8.17 Own Consumption Optimisation***.

5.3.3.10 Energy Traffic Light

Property	Recommended connections		
Serves to display the independence of the last 30 days	Independence	Independence rating	

Templates are stored in the folder ***SENEC Samples*** in -> Catalog -> Library. For more detailed information, please refer to chapter ***3.5 The Library***.

Note:	Depending on the system, the values displayed in the visualization may differ from the values displayed in the SENEC portal.

5.4 Picture Selection

Installing JUNG Visu Pro creates a new design directory. This graphic library contains all graphics used in the visualization such as backgrounds, buttons and icons.

All used graphics are copied in the project directory and the paths are converted to symbolic paths. This effects that all graphics of the project are integrated in the control panel during export.

That way the path specifications do not have to be reconfigured if the project has been imported to a different computer. Visu Pro must also be installed on the target system.

5.4.1 Symbolic Path: @sys

In den following example you can see the settings of a switch. Here you can select symbols for different states.

*The graphic for **off** state is located in the design directory under:*
c:\Users\Public\Documents\JUNG Visu Pro/App/VisSysResources/Design/images/fap/icons/licht_aus.png

*The resulting symbolic path is: **@sys/images/fap/icons/licht_aus.png***

Graphics with a path from this directory structure remain in the design directory and can be found at the same position in the target system.

5.4.2 Symbolic Path: @proj

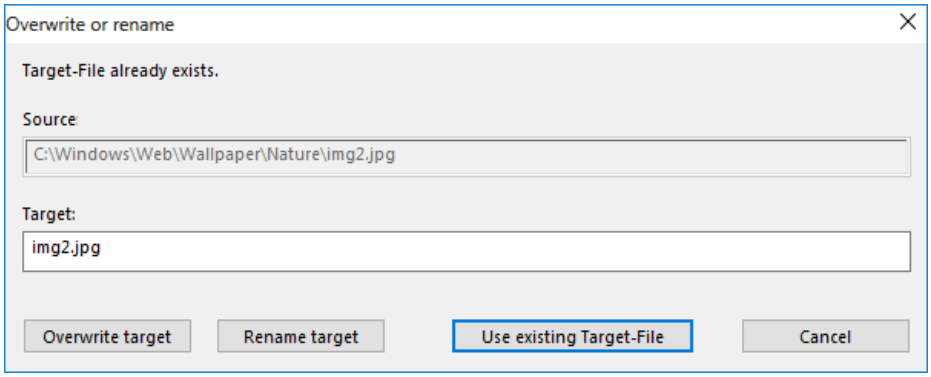
The following example shows the selection of a picture, which is not located in the JUNG Visu Pro design directory.

The selected picture is located in the directory under:
c:/WINDOWS/Web/Wallpaper/Nature/img2.jpg
*After you have confirmed the dialog with **Open**, the picture is copied in the project directory of the current visualization project.*
c:\Users\Public\Documents\JUNG Visu Pro\Stations\local\projects\<PROJECT-NAME.VIS>/images/img2.jpg

*The resulting symbolic path is: **@proj/images/img2.jpg***

Graphics with a path from this directory structure are located in the project directory and are also imported on the target system.

If a graphic with the same name already exists in the project directory, the following dialog opens:



Overwrite target: The current graphic overwrites the existing file.

Rename target: After renaming the current graphic, it is copied with a new name into the project directory and it is used.

Use existing Target-File: The copy process isn't performed and the existing graphic is used.

5.4.3 Animated Images

It is possible to set an animated image for the **background** of the following elements:

Dimmer, Blind, Slat, Slider, multipurpose control, Multipurpose dispay, Control send fixed value, Control step value, Setpoint and Setpoint shifting.

The single images get automatically extracted form the visualization editor. In the server, the appropriate image is displayed according to the received value.

Note: Only animated images in gif format are currently supported.

JUNG Visu Pro Server

Content:

1 TECHNICAL REQUIREMENTS....2

1.1 Hardware....2
1.2 Operating Systems....2
1.3 Software....2
1.4 Client/Browser Settings....2

2 FIRST RUN....3

3 ADMINISTRATION SURFACE....3

3.1 Remote Access....4
3.2 Project Settings....7
3.3 User Management....8
3.3.1 Connection Limit....8
3.3.2 Edit User Settings....9
3.3.3 Disconnect User....11
3.3.4 Add User....11
3.4 Import and Export of Project Settings....12
3.5 Change Administrator Password....13
3.6 User-specific Start Page....14

4 OPEN PROJECT....15

5 FURTHER INFORMATION....16

5.1 Resolution Selection and Scaling....16
5.2 Installation Note....17
5.3 Exit Station manually....19
5.4 Open a Port in the Windows Defender Firewall....20

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2020 ***Agentilo GmbH***
All Rights reserved

Trademarks

Microsoft®, DirectX®, .NET Framework®, Windows®, Windows NT® are registered trademarks of Microsoft Corporation.
All trademarks and registered trademarks are the property of their respective owners.

1 Technical Requirements

1.1 Hardware

Processor	Pentium IV or equal, running at 2,0 GHz.
Main memory	1024 MB
Free disk space (additional disc space requirements depend on the configured archives)	40 GB
Screen resolution	1024 x 768 Pixel
Colors	Color depth min. 16 bit per pixel.

1.2 Operating Systems

WINDOWS SERVER 2008 / 2012	NO
WINDOWS 10, 32 + 64-bit	YES
WINDOWS 8 and 8.1 Pro, 32 + 64-bit	NO
WINDOWS 7 Home, Professional and Ultimate, 32 + 64-bit	NO
WINDOWS XP	NO

1.3 Software

The installation of the Visu Pro with its requirements, the installation of the Microsoft Information Server 10.0 (IIS), the .NET Framework 3.5 Service Pack 1 and the .NET Framework 4.6.2.

1.4 Client/Browser Settings

- Javascript must be enabled.
- Cookies must be enabled.
- Using multiple browser tabs is browser-dependent and will not work with most browsers.
- Whether an incognito / private browsing option works, is also browser-dependent.

2 First Run

When you start the visualization server in the browser for the first time, you see the login screen. The start page is available via ***http://localhost:8083***, provided that the server was installed with the default settings.

3 Administration Surface

First, a login as an administrator is necessary to configure the visualization project, which is to be displayed. The account of the administrator only serves for the configuration of the project settings. To display a project, another user must be created.

Enter the following data:

Username: ***admin***
Password: ***0000***

Then confirm your entries with ***Login***.

NOTE:

For your own safety the preset administrator password should be changed. The newly assigned password should be stored carefully!

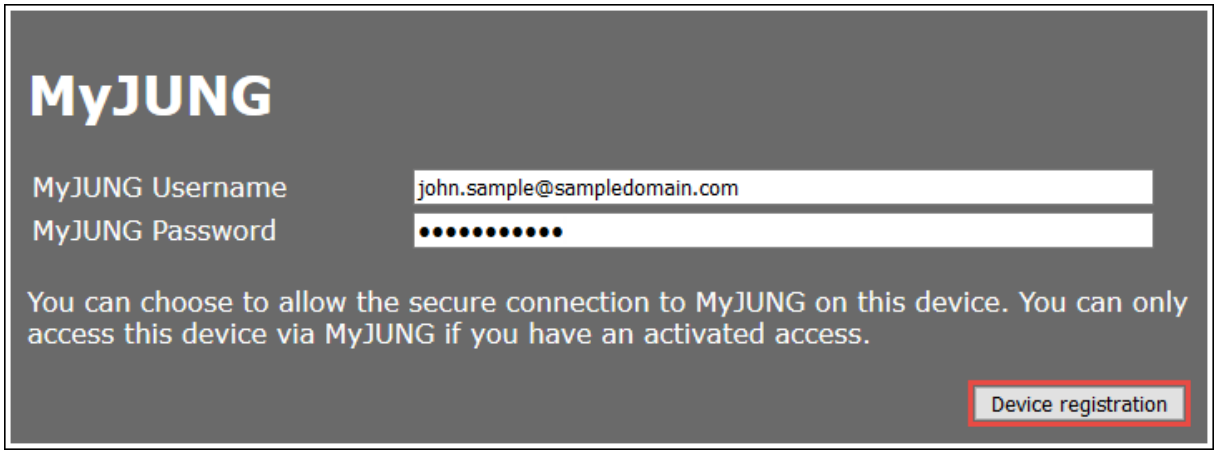
The visualization server should not be made publicly available without further security measures on the Internet.

3.1 Remote Access

A secure remote connection to the JVP visualization server can be established with the help of the chargeable remote access. This requires prior registration in the MyJUNG portal.

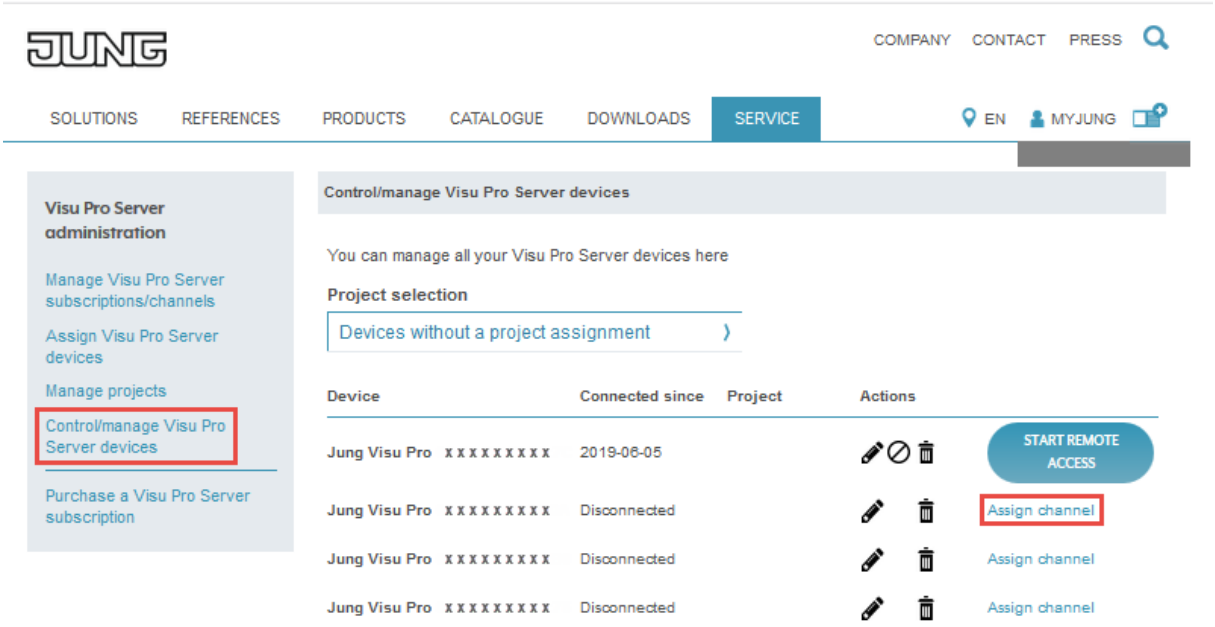
Log in to the ***admin*** or ***visuadmin*** user to configure the remote access. The default password is ***0000***.

- Enter your MyJUNG registration data in the appropriate input fields in your JVP server.
- With a click on the button ***Device registration*** your device is registered in the MyJUNG portal.



- Check the box at ***MyJUNG Connection***.

Your device is now listed in the MyJUNG Portal → Visu Pro Server administration → ***Control/manage Visu Pro Server devices***. Assign a channel to the device you just registered:



If you want to terminate remote access, click on the button ***Revoke device registration***.

If you want to restart the visualization server, click the ***Restart*** button.

By ticking the box ***Voice Services***, you enable the support of voice assistants like Alexa or Google Home for your visualization.

Before enabling the Alexa Skill or the Google Action, the JVP installation must first be registered at MyJUNG. Make sure that your visualization project is active. The visualization project can be started with the starter or by logging in to the visualization server. Then log on to the visualization server as admin or visuadmin and enter your MyJUNG data in the MyJUNG area. If the login succeeds, the field ***Voice Services*** appears in the MyJUNG area and must be activated. It can take up to one minute for the change to take effect.

The just activated JVP should now appear when you log in to the Alexa Skill or the Google Action.

3.2 Project Settings

Under ***Settings*** in the administration area you have to select a visualization project, which should be accessible by the user.

Select the desired ***visualization project*** by clicking on the drop down menu for the project. Then you can define the ***start page***, which is automatically displayed when starting the project in the browser. Finally, define the start delay in seconds and the ***resolution*** and confirm by clicking the button ***Submit & Restart.***

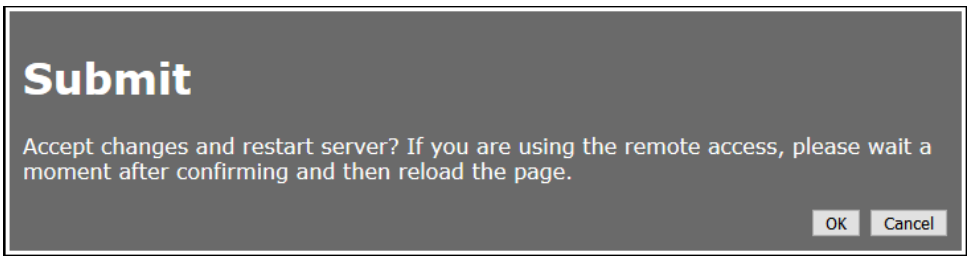
In this example, a worksheet named Start and a resolution prefix 1024x768 is searched. If no worksheet „1024x768_Start" found a worksheet is automatically selected, which is closest thereto. On the actual size of the loaded worksheet, this setting has no effect, it is used only to select the Start worksheet.

This project-related adjustment of the start worksheet is used as the default and can be overwritten in each user's settings if necessary.

About the ***Server startup delay*** you can start the visualization to the specified number of seconds delay.

The entries for the geographical position at ***Latitude*** and ***Longitude*** are important for the switching times of the Astro clock in the 16 channel weekly time switch function. By default, the values for Berlin are entered. In order for the astro clock to calculate the correct times for sunrise and sunset, the geographical position should be adjusted to the current location. The geographical position can be configured in ***admin*** as well as in ***visuadmin***.

Adopting the settings requires a server reboot. Click on ***Submit & Restart*** to apply the settings.



Then answer the confirmation prompt by clicking ***OK***. The server will then be restarted after the configured start delay.

3.3 User Management

The users list from the control panel of the Visu Pro system is listed in the administration area of the server.

Note:
The password of a pre-defined user by default is System and can be changed in the user administration of the Visu Pro.
The administration of the users is done in the control panel of the JUNG Visu Pro system. For further information, please refer to the chapter ***4.1.1 User Management*** in the help file of the JUNG Visu Pro control panel.

3.3.1 Connection Limit

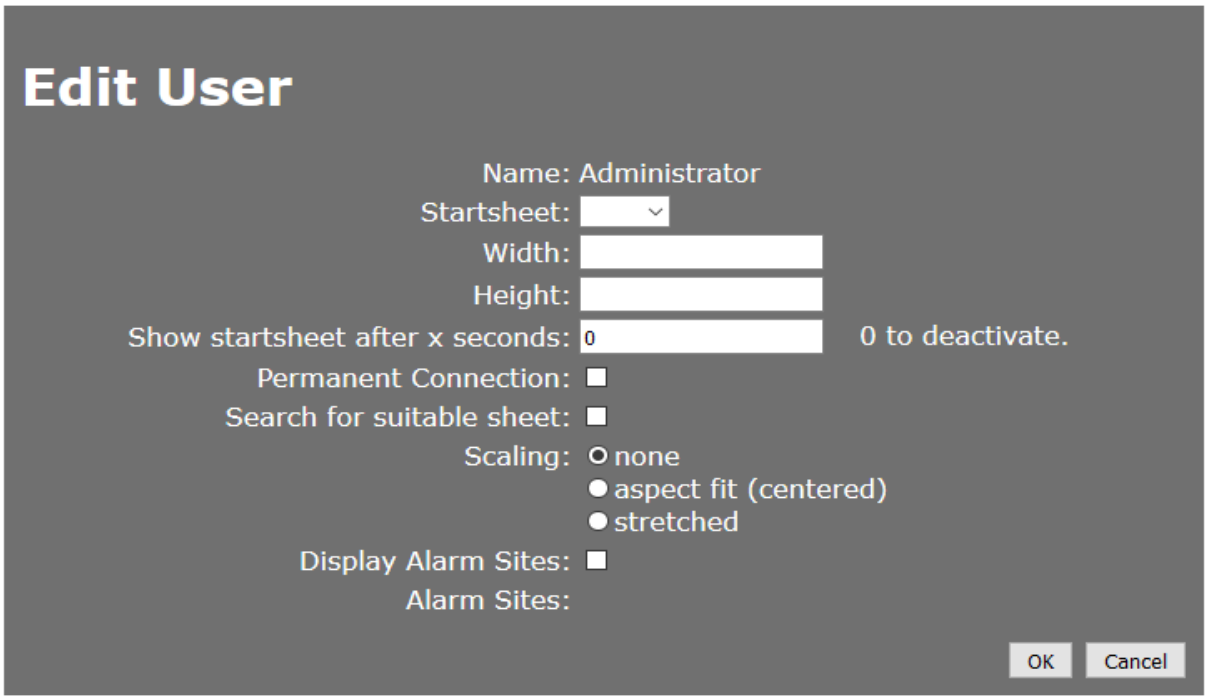
With the visualization server, maximal 25 participants/clients (hardware and operating system dependent) can connect simultaneously. If this connection limit is exceeded, the oldest inactive connection will be disconnected first. If there is no inactive connection, it will automatically disconnect the oldest active connection.

Name	Startsheet	Width	Height	Permanent	Actions
Administrator				no	Edit
Guest				no	Edit
Mainuser		1366	768	no	Edit
User				no	Edit

With the Option ***Permanent***, users can be protected from the automatic separation.

3.3.2 Edit User Settings

To edit the user settings, click in the row of the desired user the button ***Edit***.



Setting ***Startsheet***, ***Width*** and ***Height*** equal to the default settings for selecting the start worksheet and overwrites them. You have the option to assign each user a different start worksheet.

With ***Show startsheet after x seconds***, you can specify the period the home screen will be loaded, after a user is inactive. This function is not active, while an overlay (for example time switch settings) is open.

Permanent connection prevents automatic disconnect by login a new user, if all connections are exhausted to the server. These settings should be used for user accounts, what to connect Panel PCs or devices that should not be disconnected automatically (eg. JUNG SmartControl IP).

Search for suitable sheet refers to the sheet that should be visible, if the orientation changes (for example while rotating a smartphone).

- ***Active***: The server will try to find a sheet with switched resolution to the active sheet. If no suitable sheet is found, the server will search for a sheet, which is near to the switched resolution.
  Example: A sheet with the resolution 1024x768 is visible. While changing the orientation, the server will search a sheet with the resolution 768x1024. If no sheet is found the server will search for a sheet near to the resolution 768x1024.
- ***Inactive***: The server will try to find a sheet with switched resolution to the active sheet. If no suitable sheet is found, the active sheet will not be changed.

You can select a specific user scaling of the worksheet. It can be chosen between ***none, aspect fit*** and ***stretched***.

- none loads the worksheets as they were configured. They are drawn from the upper-left corner of the browser window.
- aspect fit will scale the worksheet according to the size of the browser window with a consistent aspect ratio and centered. The worksheet will not be stretched but if necessary an edge is added.
- stretched enlarges or reduces the height and width of the worksheet on the size of the browser window and scaled the content by stretching to the new size.

Enable/Disable the setting ***Display Alarm Sites*** to enable/disable automatic change to set alarm sheets.

If the setting is disabled the automatic change to alarm sheets is deactivated regardless of the settings under ***Alarm Sites***.
In the settings ***Alarm Sites*** alarm sheets for the user can be enabled or disabled individually.

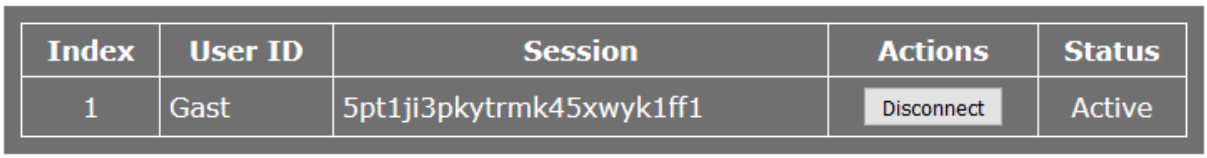
Note:
For new created users and alarm sites this option is disabled. Admin has to enable this options for each user explicitly.

In ***Edit User*** dialog submit your settings by clicking ***OK***.

Then answer the confirmation prompt by clicking ***OK***.

3.3.3 Disconnect User

The active users are displayed in the connection list.



By clicking on the ***Disconnect*** button, the connection is released.

3.3.4 Add User

The administration of the users is done in the control panel of the JUNG Visu Pro system. For further information, please refer to the chapter ***4.1.1 User Management*** in the help file of the JUNG Visu Pro control panel.

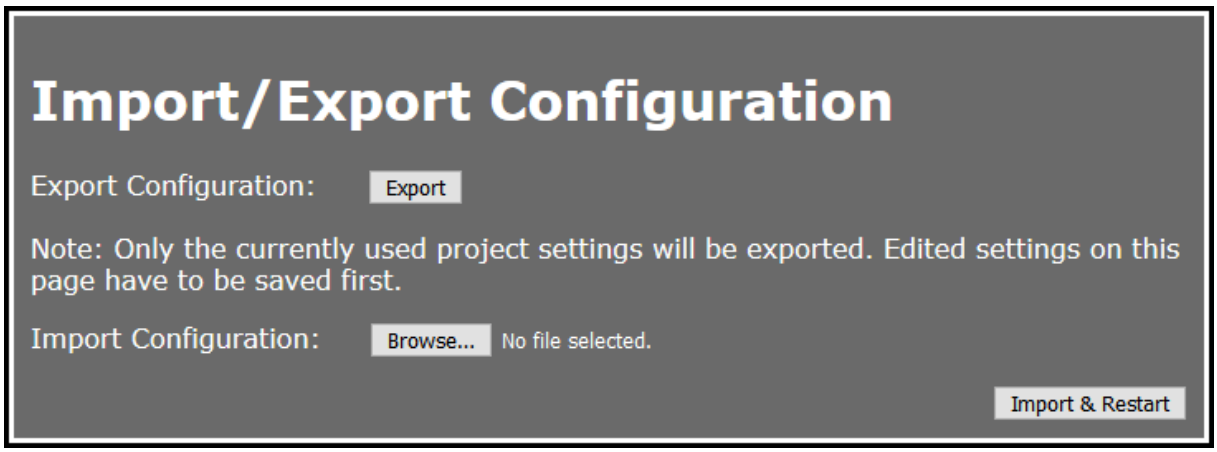
3.4 Import and Export of Project Settings

The project settings can be exported and imported to the same or a different computer.

The following settings can be exported:

- the server startup delay
- the name and startsheet of the currently active project
- the desired resolution
- all settings of all users (alarm sites, permanent connection etc.)

Note: The device registration for remote access won't be exported. This has to be executed on each computer itself.



Please note the following when **exporting**:

- a project must be set in the visualization server
- a startsheet must be set

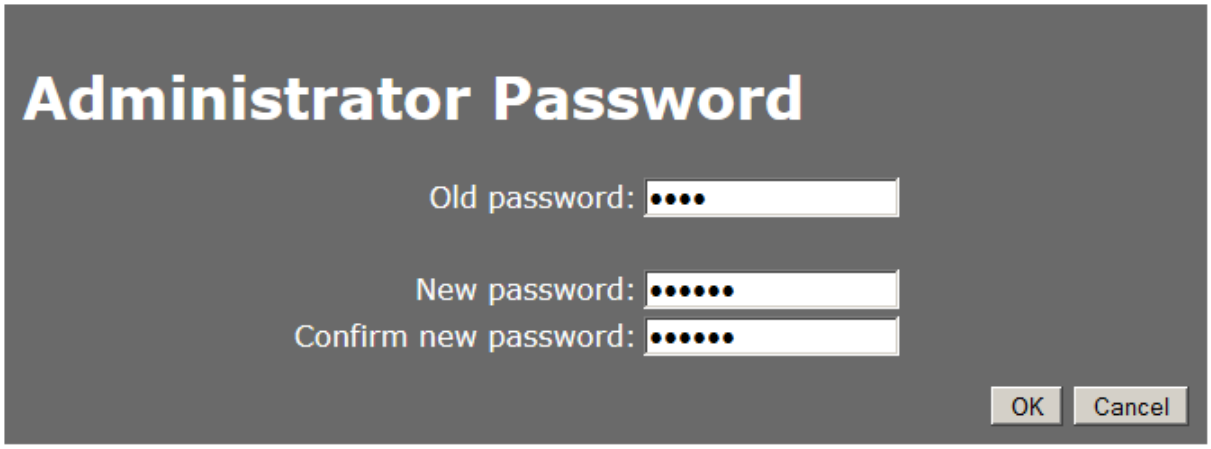
Please note the following when **importing**:

- the exported visualization project must exist on the target computer
- the exported startsheet must still exist in the visualization project of the target computer
- if the imported startsheet of a user no longer exists in the visualization project, it is ignored during the import process.
- if the exported configuration file contains users that do not exist on the target computer, they won't be imported

3.5 Change Administrator Password

To change the administrator password, click the button ***Change administrator password***.

Enter the current and the new password. Then confirm the new password in the last entry field.



Confirm the entries by clicking ***OK***.

Answer the confirmation prompt by clicking ***OK***.

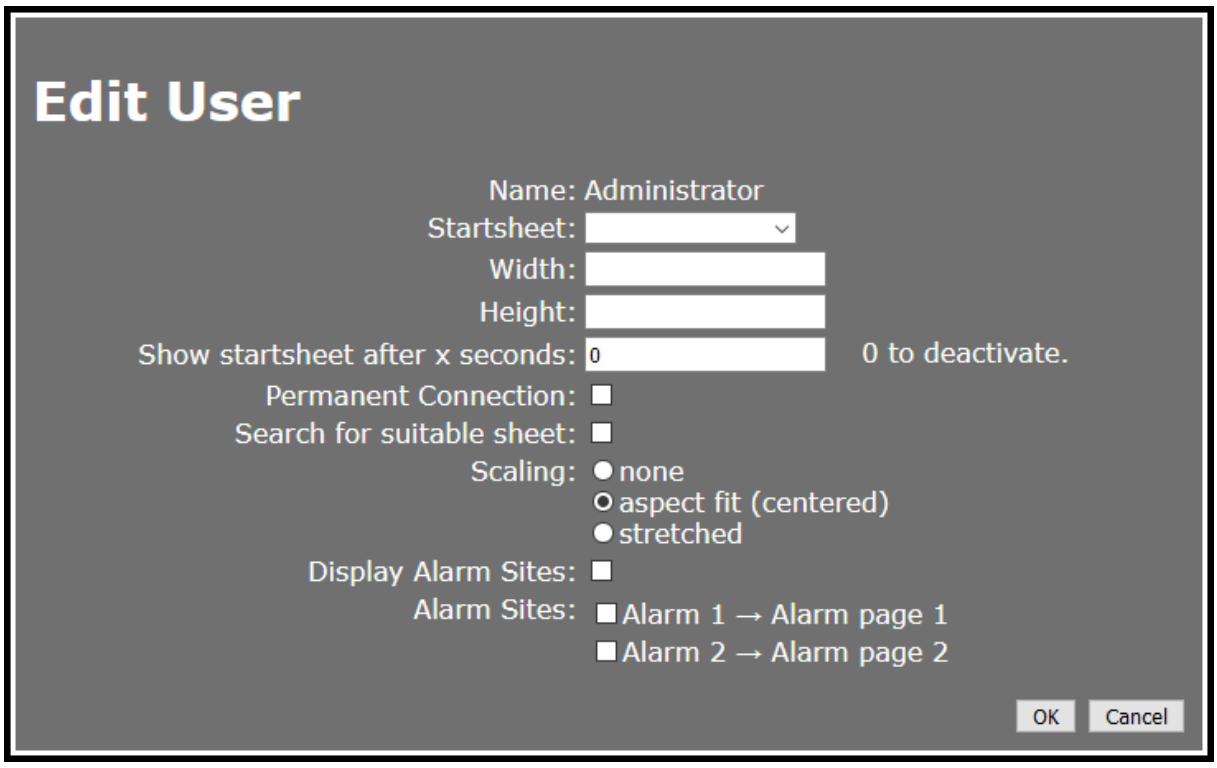
3.6 User-specific Start Page

You can assign specific start pages to individual users, that deviate from the default start page.

Enter in the address bar of the browser ***http://localhost:8083*** and press the ***Enter*** button, then log back in with the user ***admin*** and the password ***0000.***

Name	Startsheet	Width	Height	Permanent	Actions
Administrator				no	Edit
Guest				no	Edit
Mainuser		1366	768	no	Edit
User				no	Edit

Under ***User***, click in the row of the desired user on the button ***Edit***.



Select the required worksheet in the drop-down menu, configure, if applicable, the resolution and confirm by clicking ***OK***.

Answer the confirmation prompt by clicking ***OK***. The settings will then be saved and assigned to the user. Then click ***Logout*** to logout from the administration area.

After entering the user data on the login page, the assigned start page for the user is automatically displayed.

4 Open Project

Enter the user data in the login mask:

Username: *Guest*
Password: *System*

Then confirm your entries by clicking ***Login*** and the start page of the project specified before will be opened.

5 Further Information

5.1 Resolution Selection and Scaling

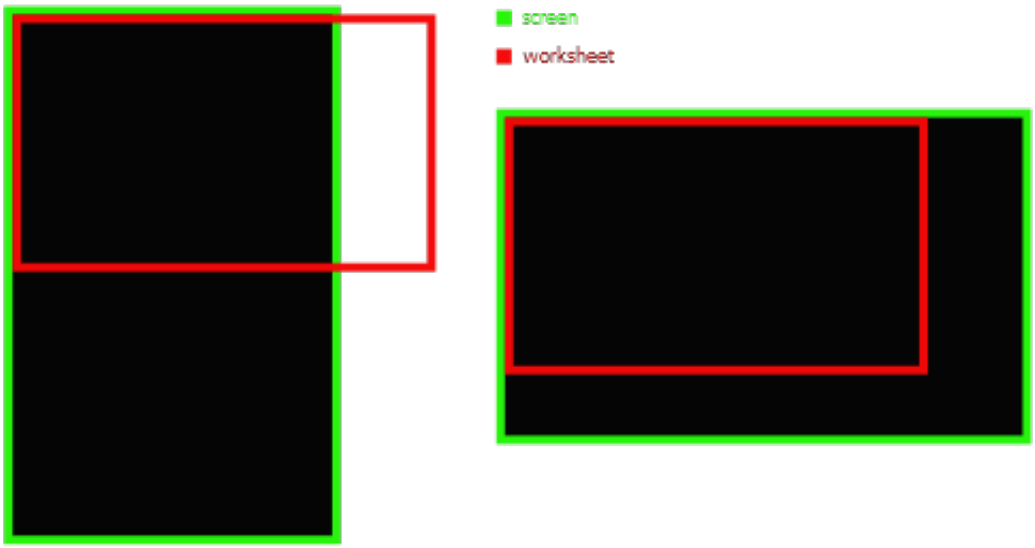
Resolution Selection

If a user has a specific worksheet size entered that doesn't exist in the project, the server choose an available worksheet resolution (for example: for an entered 1024x768 resolution with the two existing resolutions 1280x1080 & 1920x1080, the first one will be used).

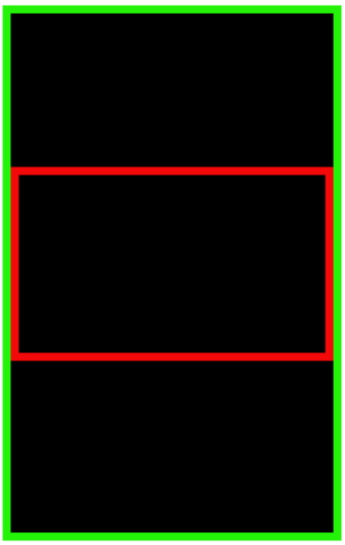
Scaling

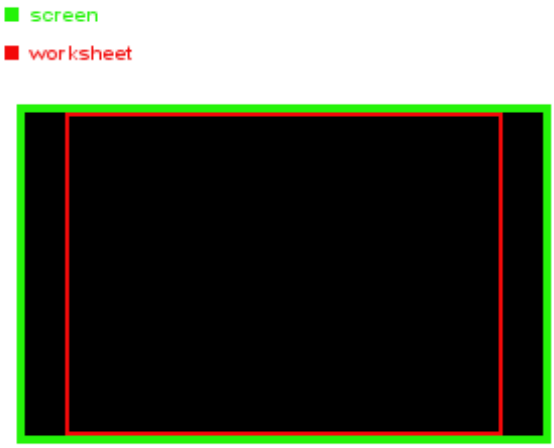
Since Visu Pro Server Version 2.3.0.9 the server automatically scales worksheets to the height and width of the screen. If the screen changes from portrait to landscape or landscape to portrait orientation (e.g. rotation iPad) and no proper worksheet is found, the current loaded sheet will be scaled to new screen width.

None

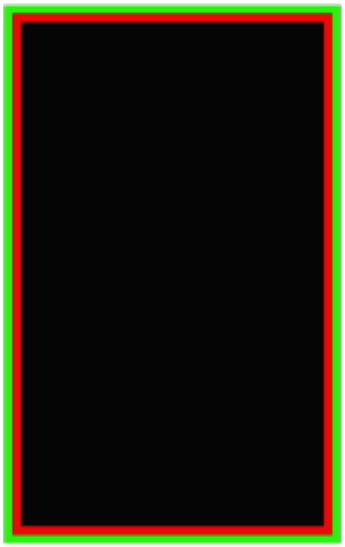


Aspect fit (centered)





Stretched

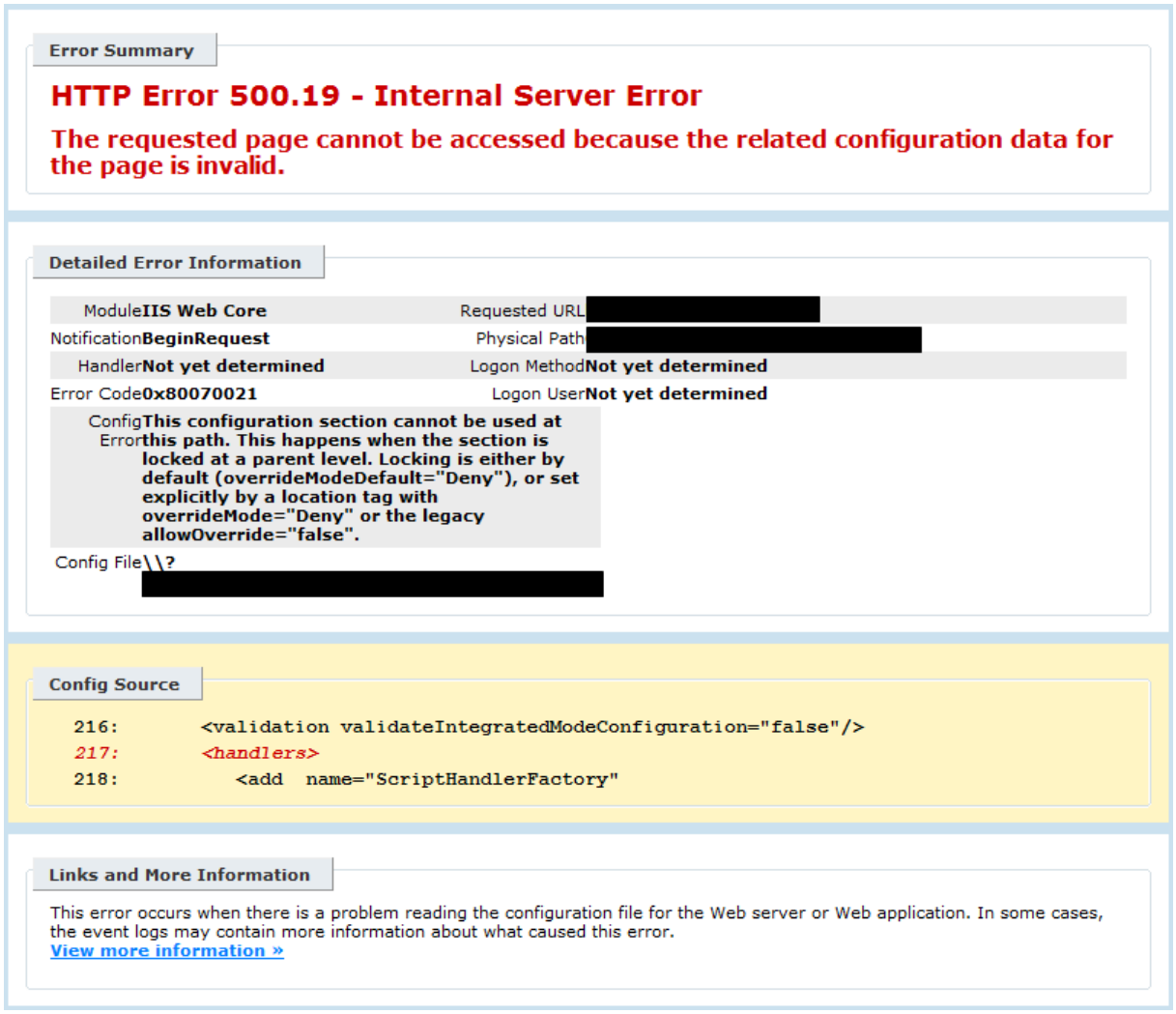


5.2 Installation Note

If you first uninstall and later reinstall the Visu Pro visualization, you must uninstall and reinstall the visualization server (after the reinstallation of the Visu Pro) as well.

For the installation and uninstallation of the visualization server run the setup. Follow the installation instructions. Please note that the Microsoft Information Server 7.5 (IIS) must be stopped before this process.

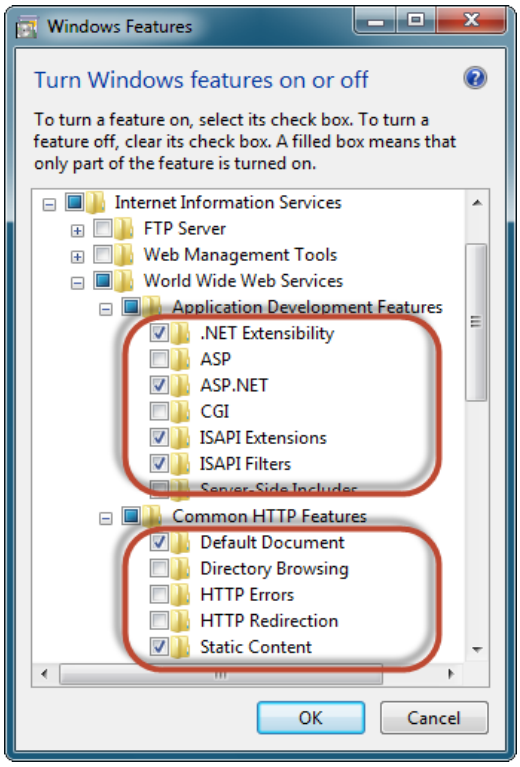
If not all the required components of IIS are installed or enabled, an error message may possibly be generated. For example:



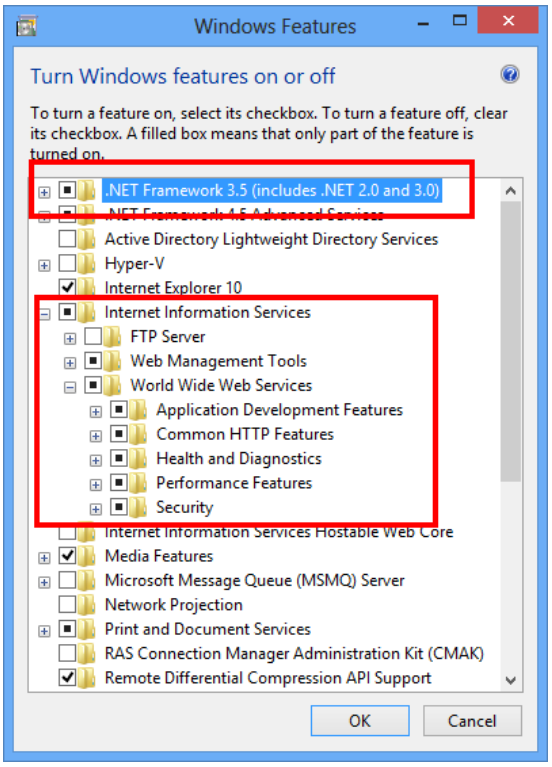
In this case, you have the option to install the required ASP.NET components.

Select Components and Functions

Select: Control panel > Programs and features > turn Windows features on or off.



Windows 7



Windows 8 / 8.1 / 10

Activate and confirm the required options under Internet Information Services > World Wide Web Services > Application Development Features > ***ASP.NET*** and Common HTTP Features > ***Static Content***. Then the activated check boxes should look like in the picture above.

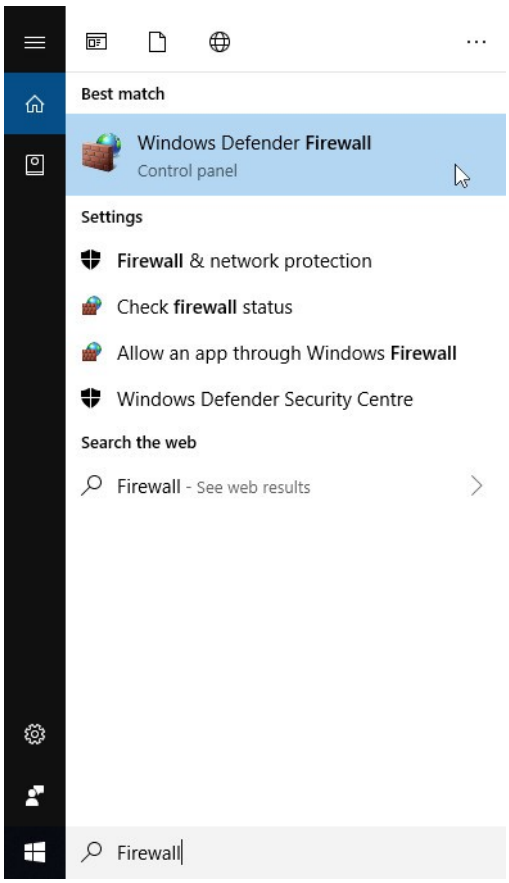
Note: Since Windows 8.1, the required **.NET Framework 3.5** is no longer delivered as automatically activated. This feature must be activated manually via the control panel in any case before the installation, because the Visu Pro Smart Assistant setup does not.

5.3 Exit Station manually

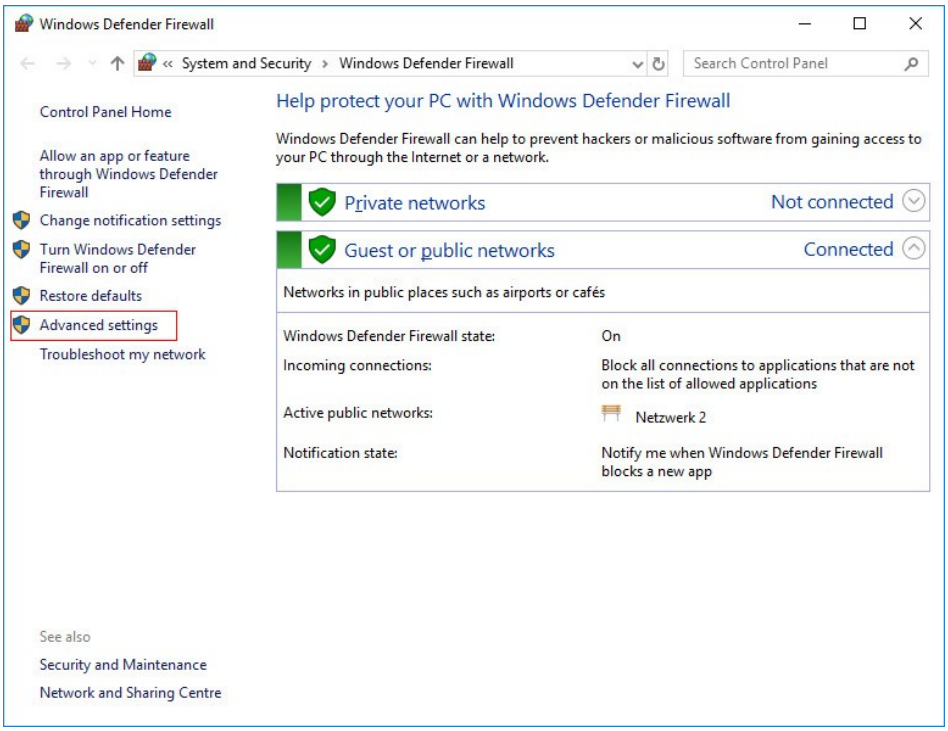
The visualization server interacts with the Visu Pro System via a component, which is named ***Station***. It is the common link between the Visu Pro modules. If the station is not already running, it is activated and started in the information area when the server is started by opening the user interface in the browser. The station continues running until the application pool is stopped explicitly in the Microsoft Information Server 7.5 (IIS).

5.4 Open a Port in the Windows Defender Firewall

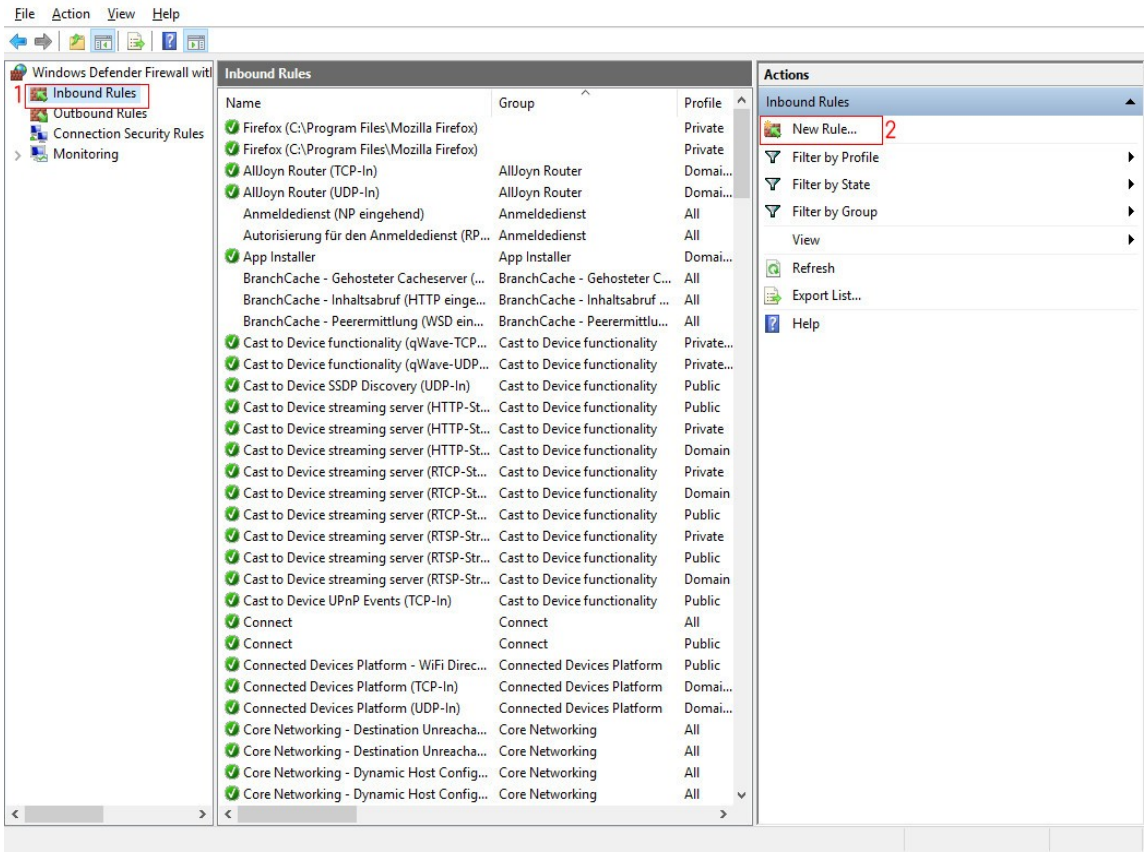
If the computer running the visualization server is using the Windows Defender Firewall, it is necessary to open up the port the server is listening to, so other devices in the local network are able to reach it.
To open a port, you have to open the Windows Defender Firewall. The easiest way to do so is to type “Firewall” into the searchbar in your Windows taskbar.



Click on the ***Advanced Settings*** menu item on the left side of the window.



The window ***Windows Defender Firewall with Advanced Security*** will open. Now select the node ***Inbound Rule*** in the left selection tree and then click onto ***New Rule…*** in the actions menu on the right.



This will open the ***New Inbound Rule Wizard***, which will lead you through five steps to create a new rule.

1. **Rule Type**:
   Choose the type **Port**.
2. **Protocol and Ports**:
   Leave the settings for the protocol as **TCP** and **Specific local ports**.
   Enter the port, on which your visualization server is listening, into the text field. The default value is 8083.
3. **Action**:
   Leave the settings as **Allow the connection**.
4. **Profile**:
   Change the settings here to fit your needs. If you are not sure, leave them as they are.
5. **Name**:
   In this step you can determine a name and a description for the new rule. We recommend that you assign a meaningful name if you want to find the rule again afterwards. An example could be **JUNG Visu Pro**.

Click on the button **Finish** afterwards. Your rule will now be displayed in the list of inbound rules in the window ***Windows Defender Firewall with Advanced Security*** and therefore your JUNG Visu Pro visualization server is now reachable in your local network.

JUNG Visu Pro
Starter

Content:

1 INTRODUCTION.....2

2 USER INTERFACE.....2

2.1 The Menu.....2

3 FUNCTIONS.....3

3.1 Select Project.....3
3.2 Start and Stop Projects manually.....4
3.3 Start Project automatically.....5
3.4 Update Project Lists.....5
3.5 IIS Reset.....6

Address

ALBRECHT JUNG GMBH & CO. KG
Volmestraße 1
D-58579 Schalksmühle
Federal Republic of Germany

Phone +49 (0) 23 55/80 60
Fax +49 (0) 23 55/80 61 89

mail.info@jung.de

Copyright ©2019 ***Agentilo GmbH***
All Rights reserved

Trademarks

EIB®, KNX® and ETS® are registered trademarks of the KNX Association.
LON® is a registered trademark of Echelon Corporation registered in the United States and other countries.
OPC® is a registered trademark of OPC Foundation.
Sax Basic Engine is a trademark of Sax Software Corporation.
Adobe Acrobat® is a registered trademark of Adobe Systems Incorporated.
Microsoft®, ActiveX®, DirectX®, Windows®, Windows NT®, Excel®, Visual Basic® are registered trademarks of Microsoft Corporation.

All trademarks and registered trademarks are the property of their respective owners.

1 Introduction

After the visualization system has been installed, a program group is created in the Start Menu. The default settings can be adopted or manually changed during installation.

To start the control panel of the visualization, click the program icon. In order to clearly distinguish between the different applications and to facilitate the navigation of the system, different colors have been assigned to the individual application icons. The icon of the **Starter** contains a green colored bar .

To open the function modules, double-click the respective module icon.

The Visu Pro Starter is an application that serves to start and stop a process model of the Visu Pro Visualization System. Thus, the process model is run independently from the Visualization Server and from the users that might be logged in there.

The Starter is able to make an entry into the Registry in order to start automatically after the computer is booted. This setting can be activated for the currently logged in user or for all users of the system.

After the configured delay time has elapsed, the selected projects will be started. The process model must be connected to the configured visualization project in the Visualization Server.

Proceeding:

1. Open the Starter.
2. Select a process model from the project list.
3. Set the autostart options and the delay times in seconds.
4. Confirm the entries by clicking the button ***Save settings***.
5. To start the projects, click the button ***Start***.

2 User Interface

2.1 The Menu

Menu item *File*

Exit: Closes the editor. Shortcut ALT + F4.

Menu item Language

To choose a language, select the desired entry from menu.

Menu item *Help*

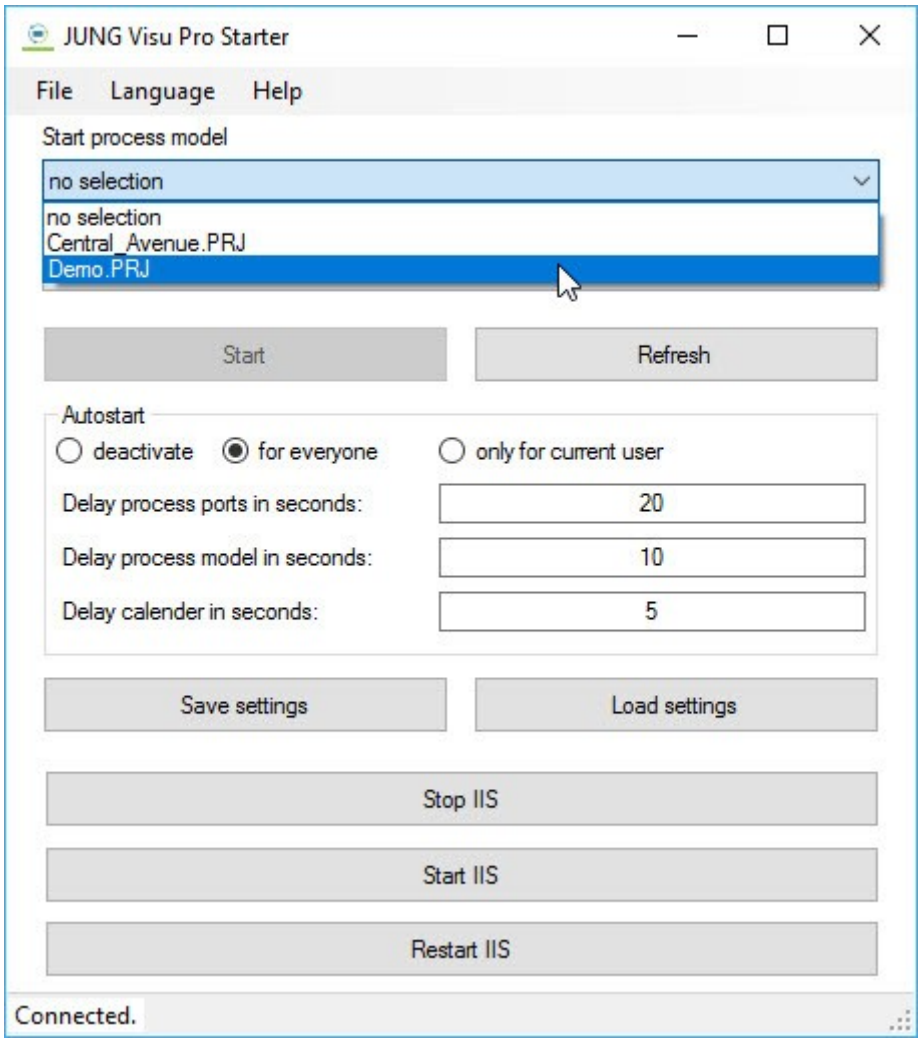
Help: Opens the help file. Shortcut F1

Version: Contains version and copyright information of this program.

3 Functions

3.1 Select Project

To choose a project, select the desired process model from the project list.

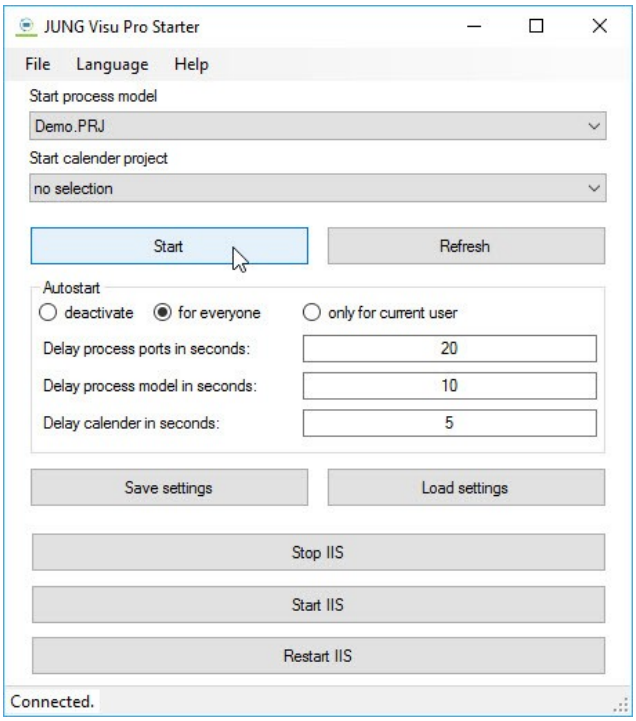


Set the autostart options. The settings can be activated for the currently logged in user or for all users of the system. Then set the time delays in seconds.

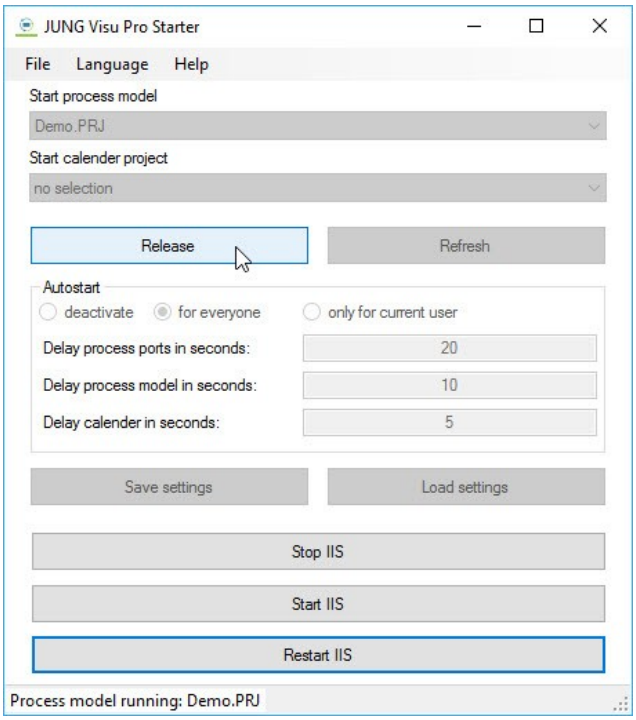
Confirm the settings by clicking the button ***Save settings***.

3.2 Start and Stop Projects manually

After the settings have been saved, click the button ***Start*** to start the selected projects.

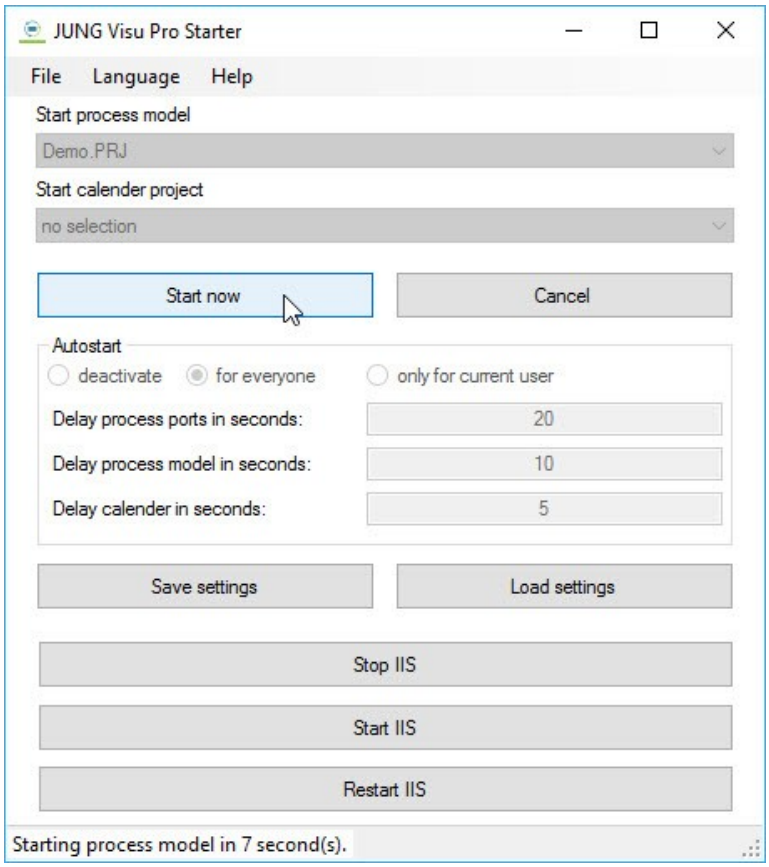


The button ***Release*** unblocks the process model again by the Visu Pro Starter. If the project is currently not in use by the server or the process model editor, it will be stopped by the release. The Starter settings can then be edited.



3.3 Start Project automatically

After the computer is rebooted, the Visu Pro Starter is opened automatically. The selected projects are opened after the configured time delays have elapsed.



By clicking the button ***Start now*** the projects can be started before the configured time delay has elapsed.

By clicking the button ***Cancel*** the starting will be cancelled.

If you start the Starter manually via the Control Panel of the Visu Pro System, the configured projects will not be started automatically. In this case, press the button ***Start*** to open the selected projects.

3.4 Update Project Lists

When the Visu Pro Starter is launched, the project lists are re-read. Newly created projects are automatically displayed in the project selection lists. However, if the Starter is already opened, the projects will not be automatically re-read.

Click the button ***Stop*** if a process model is active. Then the button ***Update*** will be activated. Click it to update the project lists.

3.5 IIS Reset

The button serves the *Internet Information Service* to restart under which the *Visu Pro Server* is running.
By starting a project it will be locked for further editing. To free up the locked project (for example, to make changes with the visualization editor) the IIS must be restarted.

Data privacy statement

1. An overview of data protection

General notes
The following notes provide a simple overview of what happens to your personal data when you visit our website. Use of the internet pages of Albrecht Jung GmbH & Co. KG is possible in principle without providing any personal data. If an individual would like to make use of a particular service offered by our company via our internet page, the processing of personal data could however become necessary. If the processing of personal data is necessary and there is no legal basis for this type of processing, we generally seek the approval of the individual. Personal data denotes all data with which you can be personally identified. Detailed information about the topic of data protection can be taken from our data protection declaration listed below.

Data collection on our website
Who is responsible for the data collection on this website?
The data collected on this website are processed by the website operator. The operator's contact details can be found in the website's required legal notice.
How do we collect your data?
Some data are collected when you provide it to us. This could, for example, be data you enter on a contact form.
Other data are collected automatically by our IT systems when you visit the website. These data are primarily technical data such as the browser and operating system you are using or when you accessed the page. These data are collected automatically as soon as you enter our website.

What do we use your data for? Part of the data is collected to ensure the proper functioning of the website. Other data can be used to analyse how visitors use the site.

What rights do you have regarding your data? You always have the right to request information about your stored data, its origin, its recipients, and the purpose of its collection at no charge. You also have the right to request that it be corrected, blocked, or deleted. You can contact us at any time using the address given in the legal notice if you have further questions about the issue of privacy and data protection. You may also, of course, file a complaint with the competent regulatory authorities.
Analytics and third-party tools
When visiting our website, statistical analyses may be made of your surfing behaviour. This happens primarily using cookies and analytics. The analysis of your surfing behaviour is usually anonymous, i.e. we will not be able to identify you from this data. You can object to this analysis or prevent it by not using certain tools. Detailed information can be found in the following privacy policy.
You can object to this analysis. We will inform you below about how to exercise your options in this regard.

2. General information and mandatory information

Data protection
The operators of this website take the protection of your personal data very seriously. We treat your personal data as confidential and in accordance with the statutory data protection regulations and this privacy policy.
If you use this website, various pieces of personal data will be collected. Personal information is any data with which you could be personally identified. This privacy policy explains what information we collect and what we use it for. It also explains how and for what purpose this happens.
Please note that data transmitted via the internet (e.g. via email communication) may be subject to security breaches. Complete protection of your data from third-party access is not possible.
Notice concerning the party responsible for this website
The party responsible for processing data on this website is:
Albrecht JUNG GmbH & Co. KG

P.O. Box 1320
58569 Schalksmühle

Correspondence address
Volmestraße 1
58579 Schalksmühle
Tel: 02355/806-0
Email: kundencenter@jung.de
The responsible party is the natural or legal person who alone or jointly with others decides on the purposes and means of processing personal data (names, email addresses, etc.).
Revocation of your consent to the processing of your data
Many data processing operations are only possible with your express consent. You may revoke your consent at any time with future effect. An informal email making this request is sufficient. The data processed before we receive your request may still be legally processed.
Right to file complaints with regulatory authorities

If there has been a breach of data protection legislation, the person affected may file a complaint with the competent regulatory authorities. The competent regulatory authority for matters related to data protection legislation is the data protection officer of the German state in which our company is headquartered.

The competent authority and the contact data are as follows:

State officer for data protection and freedom of information
Nordrhein-Westfalen
P.O. Box 20 04 44
40102 Düsseldorf

Tel.: 0211/38424-0
Fax: 0211/38424-10
Email: poststelle@ldi.nrw.de

SSL or TLS encryption
This site uses SSL or TLS encryption for security reasons and for the protection of the transmission of confidential content, such as the inquiries you send to us as the site operator. You can recognise an encrypted connection in your browser's address line when it changes from "http://" to "https://" and the lock icon is displayed in your browser's address bar.
If SSL or TLS encryption is activated, the data you transfer to us cannot be read by third parties.
Encrypted payments on this website
If you enter into a contract which requires you to send us your payment information (e.g. account number for direct debits), we will require this data to process your payment.
Payment transactions using common means of payment (Visa/MasterCard, direct debit) are only made via encrypted SSL or TLS connections. You can recognise an encrypted connection in your browser's address line when it changes from "http://" to "https://" and the lock icon in your browser line is visible.
In the case of encrypted communication, any payment details you submit to us cannot be read by third parties.
Information, blocking, deletion
As permitted by law, you have the right to be provided at any time with information free of charge about any of your personal data that is stored as well as its origin, the recipient and the purpose for which it has been processed. You also have the right to have this data corrected, blocked or deleted. You can contact us at any time using the address given in our legal notice if you have further questions on the topic of personal data.

Opposition to promotional emails We hereby expressly prohibit the use of contact data published in the context of website legal notice requirements with regard to sending promotional and informational materials not expressly requested. The website operator reserves the right to take specific legal action if unsolicited advertising material, such as email spam, is received.

3. Data protection officer

Statutory data protection officer
We have appointed a data protection officer for our company. The data protection officer for Albrecht Jung GmbH & Co.KG can be reached at the following corresponding address or email address:
Albrecht Jung GmbH & Co.KG
Data Protection Department, for the attention of Mr Schröder
Volmestraße 1
58579 Schalksmühle
Email address: datenschutz@jung.de

4. Data collection on our website

Cookies
Some of our web pages use cookies. Cookies do not harm your computer and do not contain any viruses. Cookies help make our website more user-friendly, efficient, and secure. Cookies are small text files that are stored on your computer and saved by your browser.
Most of the cookies we use are so-called "session cookies." They are automatically deleted after your visit. Other cookies remain in your device's memory until you delete them. These cookies make it possible to recognise your browser when you next visit the site.
You can configure your browser to inform you about the use of cookies so that you can decide on a case-by-case basis whether to accept or reject a cookie. Alternatively, your browser can be configured to automatically accept cookies under certain conditions or to always reject them, or to automatically delete cookies when closing your browser. Disabling cookies may limit the functionality of this website.
Cookies which are necessary to allow electronic communications or to provide certain functions you wish to use (such as the shopping cart) are stored pursuant to Art. 6 paragraph 1, letter f of DSGVO. The website operator has a legitimate interest in the storage of cookies to ensure an optimised service provided free of technical errors. If other cookies (such as those used to analyse your surfing behaviour) are also stored, they will be treated separately in this privacy policy.
Server log files
The website provider automatically collects and stores information that your browser automatically transmits to us in "server log files". These are:

- Browser type and browser version
- Operating system used
- Referrer URL
- Host name of the accessing computer
- Time of the server request

IP address

These data will not be combined with data from other sources.

The basis for data processing is Art. 6 (1) (f) DSGVO, which allows the processing of data to fulfil a contract or for measures preliminary to a contract.

Registration on this website

You can register on our website in order to access additional functions offered here. The input data will only be used for the purpose of using the respective site or service for which you have registered. The mandatory information requested during registration must be provided in full. Otherwise, we will reject your registration.

To inform you about important changes such as those within the scope of our site or technical changes, we will use the email address specified during registration.

We will process the data provided during registration only based on your consent according to Art. 6 (1)(a) DSGVO. You may revoke your consent at any time with future effect. An informal email making this request is sufficient. The data processed before we receive your request may still be legally processed.

We will continue to store the data collected during registration for as long as you remain registered on our website. Statutory retention periods remain unaffected.

Processing of data (customer and contract data)

We collect, process, and use personal data only insofar as it is necessary to establish, or modify legal relationships with us (master data). This is done based on Art. 6 (1) (b) DSGVO, which allows the processing of data to fulfil a contract or for measures preliminary to a contract. We collect, process and use your personal data when accessing our website (usage data) only to the extent required to enable you to access our service or to bill you for the same.

Collected customer data shall be deleted after completion of the order or termination of the business relationship. Legal retention periods remain unaffected.

Data transmitted when entering into a contract with online shops, retailers, and mail order

We transmit personally identifiable data to third parties only to the extent required to fulfil the terms of your contract, for example, to companies entrusted to deliver goods to your location or banks entrusted to process your payments.Your data will not be transmitted for any other purpose unless you have given your express permission to do so. Your data will not be disclosed to third parties for advertising purposes without your express consent.

The basis for data processing is Art. 6 (1) (b) DSGVO, which allows the processing of data to fulfil a contract or for measures preliminary to a contract.

Data transferred when signing up for services and digital content

We transmit personally identifiable data to third parties only to the extent required to fulfil the terms of your contract with us, for example, to banks entrusted to process your payments.

Your data will not be transmitted for any other purpose unless you have given your express permission to do so. Your data will not be disclosed to third parties for advertising purposes without your express consent.

The basis for data processing is Art. 6 (1) (b) DSGVO, which allows the processing of data to fulfil a contract or for measures preliminary to a contract.

5. Social media

Facebook plugins (Like & Share buttons)

Our website includes plugins for the social network Facebook, Facebook Inc., 1 Hacker Way, Menlo Park, California 94025, USA. The Facebook plugins can be recognised by the Facebook logo or the Like button on our site. For an overview of Facebook plugins, see https://developers.facebook.com /docs/plugins/.

When you visit our site, a direct connection between your browser and the Facebook server is established via the plugin. This enables Facebook to receive information that you have visited our site from your IP address. If you click on the Facebook "Like button" while you are logged into your Facebook account, you can link the content of our site to your Facebook profile. This allows Facebook to associate visits to our site with your user account. Please note that, as the operator of this site, we have no knowledge of the content of the data transmitted to Facebook or of how Facebook uses these data. For more information, please see Facebook's privacy policy at https://de-de.facebook.com/policy.php.

If you do not want Facebook to associate your visit to our site with your Facebook account, please log out of your Facebook account.

Twitter plugin Functions of the Twitter service have been integrated into our website and app. These features are offered by Twitter Inc., 1355 Market Street, Suite 900, San Francisco, CA 94103, USA. When you use Twitter and the “Retweet” function, the websites you visit are connected to your Twitter account and made known to other users. In doing so, data will also be transferred to Twitter. We would like to point out that, as the provider of these pages, we have no knowledge of the content of the data transmitted or how it will be used by Twitter. For more information on Twitter's privacy policy, please go to https://twitter.com/privacy.

Your privacy preferences with Twitter can be modified in your account settings at https://twitter.com/account/settings.

Google+ plugin

Our pages use Google+ functions. It is operated by Google Inc., 1600 Amphitheatre Parkway, Mountain View, CA 94043, USA.

Collection and disclosure of information: Using the Google +1 button allows you to publish information worldwide. By means of the Google+ button, you and other users can receive custom content from Google and our partners. Google stores both the fact that you have +1'd a piece of content and information about the page you were viewing when you clicked +1. Your +1 can be displayed together with your profile name and photo in Google services, for example in search results or in your Google profile, or in other places on websites and advertisements on the Internet.

Google records information about your +1 activities to improve Google services for you and others. To use the Google + button, you need a globally visible, public Google profile that must contain at least the name chosen for the profile. This name is used by all Google services. In some cases, this name may also replace a different name that you have used to share content via your Google account. The identity of your Google profile can be shown to users who know your email address or other information that can identify you.

Use of collected data: In addition to the uses mentioned above, the information you provide is used in accordance with the applicable Google data protection policies. Google may publish summary statistics about users' +1 activity or share it with users and partners, such as publishers, advertisers, or affiliate websites.

6. Data protection declaration for use of the JUNG VISU PRO Alexa skill

The protection of your personal data is of the highest priority for Albrecht JUNG GmbH & Co. KG (hereafter called: JUNG). This principle applies both to

our website and our conventional services.

JUNG therefore collects and processes your data in accordance with the current data protection law, in particular the provisions of the German Telemedia Act (TMG) and the German Data Protection Act (BDSG). In so doing, JUNG follows the principle of data avoidance and data economy.

All of the data collected by users is treated confidentially. JUNG expressly states that personal data is not passed onto a third party.

Further analysis of your usage data is not carried out by JUNG without your consent.

As soon as you activate the JUNG VIUS PRO skill, the following personal data is processed:

- Name, description and ID of the devices which have been released for the skill to control

Through the activation of the skill, Amazon can link the data of your Amazon account to the personal data of the skill. JUNG does not have any influence on this. Please take the data protection declaration of Amazon Alexa from this link: https://www.amazon.de/gp/help/customer/display.html?nodeId=3312401

When you agree to the data protection declaration, JUNG saves the following data to enable the identification of the connection details.

- IP address, date and time of access

All users have the option at any time to revoke the consent to the processing of your personal data for the future. To do this, however, you must disable the use of the JUNG VIUS PRO skill.

7. SV-Server remote access

You have already made some registration data available on our internet platform MyJUNG. You were already able to view the privacy statement when entering your data. Some of this data was retrieved again when configuring your system in this app. This is primarily used to make the services available that you have commissioned.
You must enter other data to communicate with the systems in your network.
We will clarify below which data we will process for which purpose and which rights you have regarding this.

Purpose of data processing
The processing of your data is necessary for the handling of contractual services. We process your data for the following purposes:

* According to article 6 paragraph I lit. b) of the EU GDPR based on the contract agreed with you:

- Completion of the contractual service (MyJUNG user name: generally your email address, password) for alignment with the MyJUNG-Portal.
- Supply of contractually ordered products and services (according to information and contract in the MyJUNG Portal).
- Implementation of technical functions (Device UUID, Device Key (for unique identification of your device)
- For safeguarding of internal communication in your network (IP address, subnet mask of your smartphone/tablet and your SV-Server, without alignment and transmission)

* According to article 6 paragraph I lit. a) of the EU GDPR based on the consent you have provided:

- Transmission to our own customer platform “MyJUNG” for comparison with contractual data and identification of your user (necessary for enabling remote access) on your SV-Server (control and administration server), checking of the acquired system licences.

* Duration of the processing:
We only process your data for the period required to complete our contract or the relevant regulations of our relationship. The technical data containing personal information is stored until you use the service (or the product).

8. Clever Config App

We do not collect any personal data in connection with the use of the “Jung Clever Config App”.

Only a statistical evaluation is collected via the app store which does not collect any personal data. We only obtain summarised evaluations which cannot be traced back to individual users and which inform us about the frequency and type of use. This anonymous and summarised data is used for statistical purposes and for the further development of the app.

9. Analytics and advertising

Google Analytics
This website uses Google Analytics, a web analytics service. It is operated by Google Inc., 1600 Amphitheatre Parkway, Mountain View, CA 94043, USA.
Google Analytics uses so-called "cookies". These are text files that are stored on your computer and that allow an analysis of the use of the website by you. The information generated by the cookie about your use of this website is usually transmitted to a Google server in the USA and stored there. Google Analytics cookies are stored based on Art. 6 (1) (f) DSGVO. The website operator has a legitimate interest in analysing user behaviour to optimise both its website and its advertising.
IP anonymisation
We have activated the IP anonymisation feature on this website. Your IP address will be shortened by Google within the European Union or other parties to the Agreement on the European Economic Area prior to transmission to the United States. Only in exceptional cases is the full IP address sent to a Google server in the US and shortened there. Google will use this information on behalf of the operator of this website to evaluate your use of the website, to compile reports on website activity, and to provide other services regarding website activity and Internet usage for the website operator.

The IP address transmitted by your browser as part of Google Analytics will not be merged with any other data held by Google.
Browser plugin
You can prevent these cookies being stored by selecting the appropriate settings in your browser. However, we wish to point out that doing so may mean you will not be able to enjoy the full functionality of this website. You can also prevent the data generated by cookies about your use of the website (incl. your IP address) from being passed to Google, and the processing of these data by Google, by downloading and installing the browser plugin available at the following link: https://tools.google.com/dlpage/gaoptout?hl=en.
Objecting to the collection of data
You can prevent the collection of your data by Google Analytics by clicking on the following link. An opt-out cookie will be set to prevent your data from being collected on future visits to this site: Disable Google Analytics.
For more information about how Google Analytics handles user data, see Google's privacy policy: https://support.google.com/analytics/answer/6004245?hl=en.
Outsourced data processing
We have entered into an agreement with Google for the outsourcing of our data processing and fully implement the strict requirements of the German data protection authorities when using Google Analytics.

Demographic data collection by Google Analytics This website uses Google Analytics' demographic features. This allows reports to be generated containing statements about the age, gender, and interests of site visitors. This data comes from interest-based advertising from Google and third-party visitor data. This collected data cannot be attributed to any specific individual person. You can disable this feature at any time by adjusting the ads settings in your Google account or you can forbid the collection of your data by Google Analytics as described in the section "Refusal of data collection".

10. Newsletter

Newsletter data
If you would like to receive our newsletter, we require a valid email address as well as information that allows us to verify that you are the owner of the specified email address and that you agree to receive this newsletter. No additional data is collected or is only collected on a voluntary basis. We only use this data to send the requested information and do not pass it on to third parties.
We will, therefore, process any data you enter onto the contact form only with your consent according to Art. 6 (1) (a) DSGVO. You can revoke consent to the storage of your data and email address as well as their use for sending the newsletter at any time, e.g. through the "unsubscribe" link in the newsletter. The data processed before we receive your request may still be legally processed.
The data provided when registering for the newsletter will be used to distribute the newsletter until you cancel your subscription when said data will be deleted. Data we have stored for other purposes (e.g. email addresses for the members area) remain unaffected.

11. Plugins and tools

YouTube
Our website uses plugins from YouTube, which is operated by Google. The operator of the pages is YouTube LLC, 901 Cherry Ave., San Bruno, CA 94066, USA.
If you visit one of our pages featuring a YouTube plugin, a connection to the YouTube servers is established. Here the YouTube server is informed about which of our pages you have visited.
If you're logged into your YouTube account, YouTube allows you to associate your browsing behaviour directly with your personal profile. You can prevent this by logging out of your YouTube account.
YouTube is used to help make our website appealing. This constitutes a justified interest pursuant to Art. 6 (1) (f) DSGVO.
Further information about handling user data, can be found in the data protection declaration of YouTube under https://www.google.de/intl/de/policies/privacy.

12. Payment service providers

PayPal
Our website accepts payments via PayPal. The provider of this service is PayPal (Europe) S.à.r.l & Cie, S.C.A. (22-24 Boulevard Royal, L-2449 Luxembourg.
If you select payment via PayPal, the payment data you provide will be supplied to PayPal based on Art. 6 (1) (a) (Consent) and Art. 6 (1) (b) DSGVO (Processing for contract purposes). You have the option to revoke your consent at any time with future effect. It does not affect the processing of data previously collected.

13. Modifications made to our privacy policy

This privacy policy is valid and has the status of May 2018.
JUNG reserves the right to modify our security and privacy arrangements as far as these modifications are induced by technological progress. In such cases, our privacy policy will also be updated accordingly. Therefore, please refer to the latest revision of our privacy policy available on our website.

JUNG Facility Pilot Software License Agreement

(The English version of the License Agreement is a translation of the original German version; only the German version is legally binding.)

Version of August 12, 2013

DURING THE INSTALLATION OF THE SOFTWARE PRODUCT YOU ARE ASKED TO ACCEPT THIS LICENSE AGREEMENT. BY CLICKING ON THE "YES" BUTTON, INSTALLING THE PRODUCT, OR USING THE SOFTWARE PRODUCT, YOU ARE CONSENTING TO BE BOUND BY THIS AGREEMENT.
IF YOU DO NOT AGREE TO ALL OF THE TERMS OF THIS AGREEMENT, DO NOT INSTALL THE SOFTWARE, DO NOT USE THIS SOFTWARE, WHEN ASKED TO ACCEPT THIS LICENSE AGREEMENT CLICK ON THE "NO" BUTTON AND THE INSTALLATION PROCESS WILL NOT CONTINUE.
IF YOU DID NOT HAVE THE OPPORTUNITY TO REVIEW THIS LICENSE, FOR EXAMPLE BECAUSE THE SOFTWARE PRODUCT HAS BEEN PRE-INSTALLED ON A DEVICE, AND YOU DO NOT ACCEPT THIS AGREEMENT, CONTACT THE TRADER WHERE YOU PURCHASED THE PRODUCT AND ASK FOR A REFUND.

Using this product you accept the terms and conditions of this License.

1. Definitions

"CUSTOMER" means the legal user of this SOFTWARE.
"SOFTWARE" means a copy of the JUNG Facility-Pilot Software with its software components and the documentation with which this Agreement is provided.
"USE" or "USING" means to access, install, download, copy or otherwise benefit from using the functionality of the SOFTWARE in accordance with the documentation.
"LICENSER" are Agentilo GmbH (Kaiserslautern, Gemany) and Albrecht Jung GmbH & Co. KG (Schalksmühle, Germany).
"MACHINE" means the computer on which the SOFTWARE is installed. If the SOFTWARE is installed in a virtual machine, then "MACHINE" means this virtual machine.

2. Activation

Unless the SOFTWARE is provided with a dongle, the purchased SOFTWARE must be activated with an activation code, which is bound to a specific MACHINE. When properties of this computer are changed, another activation code may be required. Without an activation, the functionality of the SOFTWARE will be limited to a time period and a number of program starts.
Until the purchased SOFTWARE has been completely paid by the CUSTOMER, the LICENSER reserves the right to not provide an activation code.

3. Rights

The LICENSER grants to the CUSTOMER the nonexclusive right to install the SOFTWARE on one MACHINE and to USE the SOFTWARE according to this Agreement. When purchasing the SOFTWARE, the CUSTOMER receives no other rights than USING the SOFTWARE. CUSTOMER shall not modify the SOFTWARE, reverse compile or reverse assemble all or any portion of the SOFTWARE. The SOFTWARE is protected by copyright law and trademarks. The CUSTOMER is allowed to sell the SOFTWARE as a whole. In this case, he is obliged to remove the SOFTWARE from the computer where it has been installed and to destroy all copies of the SOFTWARE.

4. Limited Warranty

The SOFTWARE is developed with great care, but in no event the LICENSER warrants that the SOFTWARE is error free or that the CUSTOMER will be able to operate the SOFTWARE without problems or interruptions. The LICENSER does not warrant for problems due to improper use of the SOFTWARE or other causes, which are out of the sphere of influence of the LICENSER.
The LICENSER and the CUSTOMER are fully aware, that according to state-of-art it is not possibly to exclude software errors for all possible use cases.
Descriptions of the software do not constitute any guarantee within the meaning of §443 fff of the German Civil Code, in particular no guarantee of characteristics.

For software provided to the CUSTOMER free of charge, the software is distributed as is, without any warranty.

The LICENSER shall first be given an opportunity to render subsequent performance within a reasonable period. This claim shall be time-barred no later than 12 months after delivery of the software. The CUSTOMER shall report defects to the LICENSER in writing without undue delay. The LICENSER can opt to rectify the problem by debugging, providing new software build or showing how to avoid the effects of the fault. The CUSTOMER shall accept a new software build unless this causes him unreasonable efforts.

Except as specified in this Agreement, all express or implied representations and warranties, including any implied warranty of merchantability, fitness for a particular purpose or non-infringement, are hereby excluded to the extent allowed by applicable law.

The user must assume the entire risk of using the SOFTWARE. In no event will the LICENSER be liable for any lost revenue, profit, or data, or for special, direct, indirect, consequential, incidental, or punitive damages, however caused and regardless of the theory of liability arising out of the use of or inability to use the SOFTWARE, even if the LICENSER has been advised of the possibility of such damages. This exclusion shall not apply where e.g. under product liability law or in cases of willful misconducts there is a mandatory liability. In no event shall the LICENSER or its suppliers' liability to the CUSTOMER, whether in contract, tort (including negligence), or otherwise, exceed the price paid by the CUSTOMER. The foregoing limitations shall apply even if the above-stated warranty fails of its essential purpose.

5. Termination

This license is effective until terminated. The CUSTOMER may terminate this license at any time by destroying all copies of the SOFTWARE including any documentation. This license will terminate immediately without notice if the CUSTOMER fails to comply with any provision of this license. Upon termination, the CUSTOMER must destroy all copies of the SOFTWARE.

6. Place of jurisdiction

Place of jurisdiction is Kaiserslautern, Germany.

7. Others

If any term or other provision of this Agreement is determined to be invalid, illegal or incapable of being enforced by any rule or law, or public policy, all other conditions and provisions of this Agreement shall nevertheless remain in full force and effect so long as the economic or legal substance of the transactions contemplated hereby is not affected in any manner materially adverse to any party. Upon such determination that any term or other provision is invalid, illegal or incapable of being enforced, the parties hereto shall negotiate in good faith to modify this Agreement so as to effect the original intent of the parties as closely as possible in an acceptable manner to the end that transactions contemplated hereby are fulfilled to the extent possible.

Nutzungslizenzvertrag für 'JUNG Facility-Pilot'

Stand vom 12. August 2013

WÄHREND DER INSTALLATION DIESES SOFTWAREPRODUKTS WERDEN SIE AUFGEFORDERT DIESE LIZENZBEDINGUNGEN ZU AKZEPTIEREN. DURCH DAS BESTÄTIGEN DER LIZENZBEDINGUNGEN PER BUTTON ODER DIE VERWENDUNG DES SOFTWAREPRODUKTS SIND SIE AN DIE LIZENZBEDINUNGEN GEBUNDEN.
WENN SIE NICHT MIT ALLEN LIZENZBEDINGUNGEN EINVERSTANDEN SIND, INSTALLIEREN SIE DIE SOFTWARE NICHT, BRECHEN SIE DIE INSTALLATION AB.
HATTEN SIE NICHT DIE GELEGENHEIT DIESE LIZENZBEDINGUNGEN ZU LESEN, ZUM BEISPIEL WEIL DIE SOFTWARE AUF EINEM GERÄT VORINSTALLIERT WAR, UND AKZEPTIEREN SIE DIE LIZENZBEDINGUNGEN NICHT, DANN NEHMEN SIE ZU IHREM HÄNDLER KONTAKT AUF, UM DAS SOFTWAREPRODUKT ZURÜCKZUGEBEN.

Mit der Nutzung dieses Softwareprodukts erkennen Sie die Bedingungen dieses Nutzungslizenzvertrages an.

1. Definitionen

LIZENZGEBER: Agentilo GmbH (Kaiserslautern, Deutschland) und Albrecht Jung GmbH & Co. KG, Schalksmühle, Deutschland)
LIZENZNEHMER: Der rechtmäßige Nutzer dieser SOFTWARE.
SOFTWARE: Eine Kopie der JUNG Facility-Pilot Software, mit den Programm-Komponenten und mit der Software-Dokumentation, die zusammen mit dieser Vereinbarung geliefert wurde.
LIZENZ: Die Nutzungslizenz für SOFTWARE.
MASCHINE: Wenn SOFTWARE in einer virtuellen Maschine installiert ist, dann ist es diese virtuelle Maschine. Anderenfalls ist es der Computer, auf dem die SOFTWARE installiert ist.

2. Aktivierung des Produkts

Außer wenn die LIZENZ zusammen mit einem Dongle erworben wurde, wird die SOFTWARE durch einen Freischaltcode aktiviert, der an die Eigenschaften einer einzelnen MASCHINE gebunden ist. Bei Änderungen der MASCHINE muss die Aktivierung wiederholt werden. Ohne die Aktivierung der SOFTWARE ist ihre Nutzung zeitlich beschränkt und ist die Anzahl der Programmstarts beschränkt. Demo-Versionen von SOFTWARE sind grundsätzlich zeitlich und in der Anzahl der Programmstarts beschränkt.
Bis zur vollständigen Bezahlung der LIZENZ behält sich LIZENZGEBER das Recht vor, für die SOFTWARE keinen Freischaltcode auszustellen.

3. Rechte

LIZENZGEBER räumt dem LIZENZNEHMER das nicht ausschließliche, für die aktivierte SOFTWARE das zeitlich unbegrenzte Recht ein, die SOFTWARE gemäß den nachstehenden Bedingungen auf einer einzelnen MASCHINE zu nutzen. Durch den Erwerb von LIZENZ erhält der LIZENZNEHMER nur das Eigentum an dem Datenträger

selbst, auf dem die SOFTWARE befindlich ist. Ein Erwerb von Rechten, die über das in diesem Vertrag geregelte Nutzungsrecht hinausgehen, ist damit nicht verbunden. LIZENZGEBER behält sich insbesondere alle Veröffentlichungs- und Verwendungsrechte vor. Die SOFTWARE darf nicht zurückentwickelt, übersetzt oder verändert werden. Für die SOFTWARE besteht ein Schutz durch Urheberrechtsgesetze und andere nationale und internationale Rechtsvorschriften. Vermerke über das Urheberrecht dürfen nicht entfernt werden und müssen auf Kopien übertragen werden.
Der LIZENZNEHMER ist berechtigt, die LIZENZ als Ganzes weiterzuverkaufen. In diesem Fall verpflichtet er sich, die SOFTWARE von der Maschine zu entfernen, auf der sie installiert war, und alle Kopien des Softwareprodukts zu zerstören.

4. Gewährleistung und Haftungsausschluss

LIZENZGEBER gewährleistet, dass die SOFTWARE ordnungsgemäß dupliziert und der Datenträger frei von Material- und Herstellungsfehlern ist, sofern die SOFTWARE auf einem Datenträger geliefert wurde. Sollte der Datenträger Fehler aufweisen oder keine ordnungsgemäße Kopie der SOFTWARE enthalten, dann erhalten Sie Ersatz.
Für die Fehlerfreiheit der SOFTWARE wird keine Gewährleistung übernommen. Die Gewährleistung erstreckt sich auch nicht auf Mängel wegen unsachgemäßer Behandlung oder andere Gründe außerhalb des Einflussbereiches von LIZENZGEBER.
Die Parteien sind sich bewusst, dass es nach dem Stand der Technik nicht möglich ist, Fehler von SOFTWARE unter allen Anwendungsbedingungen auszuschließen.
Beschreibungen der SOFTWARE stellen keine Garantie im Sinne der §§ 443 ff. BGB dar, insbesondere keine Beschaffenheitsgarantie.
Für SOFTWARE, die dem LIZENZNEHMER unentgeltlich überlassen wird, unterliegt der LIZENZGEBER nur bei Arglist einer Sachmängelhaftung.

Der LIZENZGEBER wird die vom LIZENZNEHMER gemeldeten reproduzierbaren Fehler innerhalb einer angemessenen Frist beheben. Dieser Anspruch endet 12 Monate nach Lieferung. Sachmängel hat der LIZENZNEHMER gegenüber dem LIZENZGEBER sofort schriftlich zu melden. Die Nachbesserung erfolgt nach Wahl des LIZENZGEBER durch Fehlerbeseitigung, durch Überlassung eines neuen Softwarestandes oder dadurch, dass der LIZENZGEBER Möglichkeiten aufzeigt, die Auswirkungen des Fehlers zu vermeiden. Ein neuer Softwarestand ist vom LIZENZNEHMER zu übernehmen, wenn der Aufwand für den LIZENZNEHMER vertretbar ist.

Keine Haftung für Folgeschäden:

LIZENZGEBER ist nicht für Schäden (uneingeschränkt eingeschlossen sind Schäden aus entgangenem Gewinn, Betriebsunterbrechung, Verlust von geschäftlichen Informationen oder von Daten oder aus anderem finanziellen Verlust) ersatzpflichtig, die aufgrund der Benutzung von SOFTWARE oder der Unfähigkeit, dieses Produkt zu verwenden, entstehen, selbst wenn LIZENZGEBER von der Möglichkeit eines solchen Schadens unterrichtet worden ist.

Auf jeden Fall ist die Haftung von LIZENZGEBER auf den Betrag beschränkt, den LIZENZNEHMER tatsächlich für LIZENZ bezahlt hat. Dieser Ausschluss gilt nicht für Schäden, die durch Vorsatz oder grobe Fahrlässigkeit auf Seiten von LIZENZGEBER verursacht wurden. Ebenfalls bleiben Ansprüche, die auf unabdingbaren gesetzlichen Vorschriften zur Produkthaftung beruhen, unberührt.

5. Beendigung

Dieser Vertrag und die durch ihn gewährten Rechte enden, wenn der LIZENZNEHMER eine Bestimmung dieses Vertrages nicht erfüllt.

6. Gerichtsstand

Wenn LIZENZNEHMER keine Privatperson ist, dann ist Kaiserslautern, Deutschland, der Gerichtsstand.

7. Sonstiges

Jegliche Nutzung der Programme, Dokumentationen und anderer Daten von SOFTWARE, gleichgültig, ob Sie die SOFTWARE per CD, Internet oder auf anderem Weg erhalten haben, setzt Ihr Einverständnis mit den jeweiligen Lizenzvereinbarungen voraus.

Sollte eine Bestimmung dieses Vertrages unwirksam sein, wird die Wirksamkeit der übrigen Bestimmungen davon nicht berührt. Die Parteien verpflichten sich, anstelle der unwirksamen Bestimmung eine dieser Bestimmung möglichst nahekommende wirksame Regelung zu treffen.