CHORUS

GEWISS

Smart Gateway configuration and domotic supervision

Installation manual

CONTENTS

WHAT IS THE CONFIGURATION PORTAL USED FOR?

The IoT Configurator is used to programme the Smart Gateway, configuring the basic settings, the communication between devices using different protocols, the special functions offered by the Gateway, roles and users with the relative access rights and the graphic interface of the Smart Gateway app for domotic system supervision.
To configure the KNX part, the relative ETS 4.0 (or higher - .knxproj) project must be imported, whereas the ZigBee products can be implemented directly via the IoT Configurator.

REQUISITES

The IoT Configurator is accessed from the web page https://iotconfig.gewiss.cloud, using one of the commons web browsers: Chrome, Internet Explorer, Mozilla Firefox, Microsoft Edge, Safari, etc.
An Internet connection is needed when configuring the Smart Gateway and its functions, because the configuration is saved on the cloud so it can then be distributed to all the clients connected (Smart Gateway and mobile device with the app installed).
A personal account is needed in order to use the IoT Configurator and the relative services.

GEWISS IOT CONFIGURATOR

The GEWISS IoT Configurator can be accessed via the link https://iotconfig.gewiss.cloud.
To access the services and the IoT Configurator, you must enter your credentials; if you don't have an account on the GEWISS IoT Configurator, you will have to register or make the access using the account of one of the social media listed on the page.
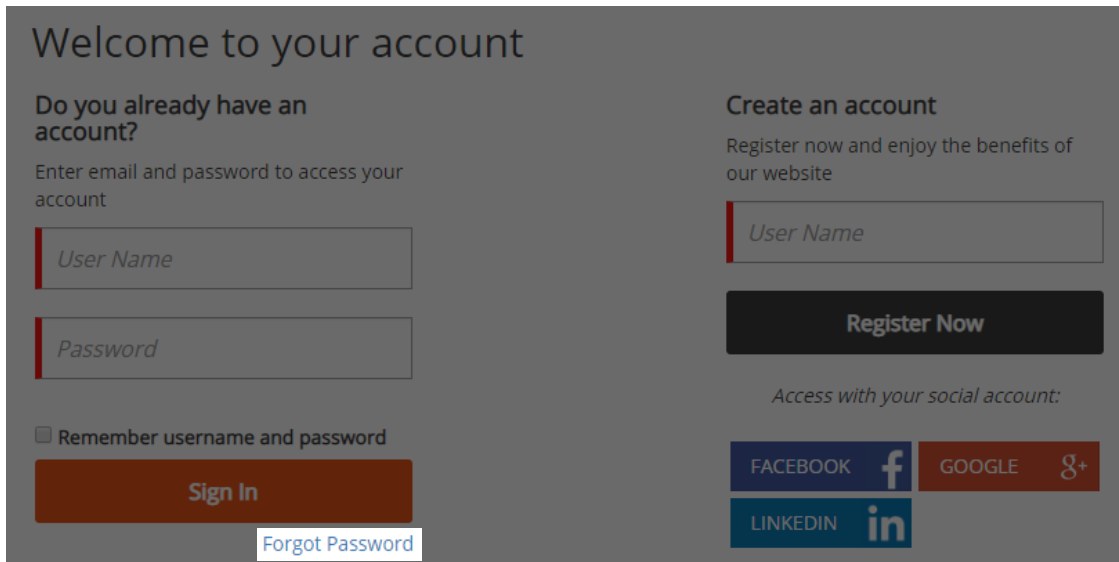
Registering on the GEWISS IoT Configurator

If you have not yet registered in the system, you have to create a new account choosing from the two options that you find in the right column of the login page. Insert you mail in the box and then click "**Register**" if you want use your mail address, or login with a social Account (Facebook, Google+, and LinkedIn.

To complete the registration, you have to fill the boxes with your information (the ones with a red label are mandatory), read and mark the information regarding the processing of personal data and press "**Register**".
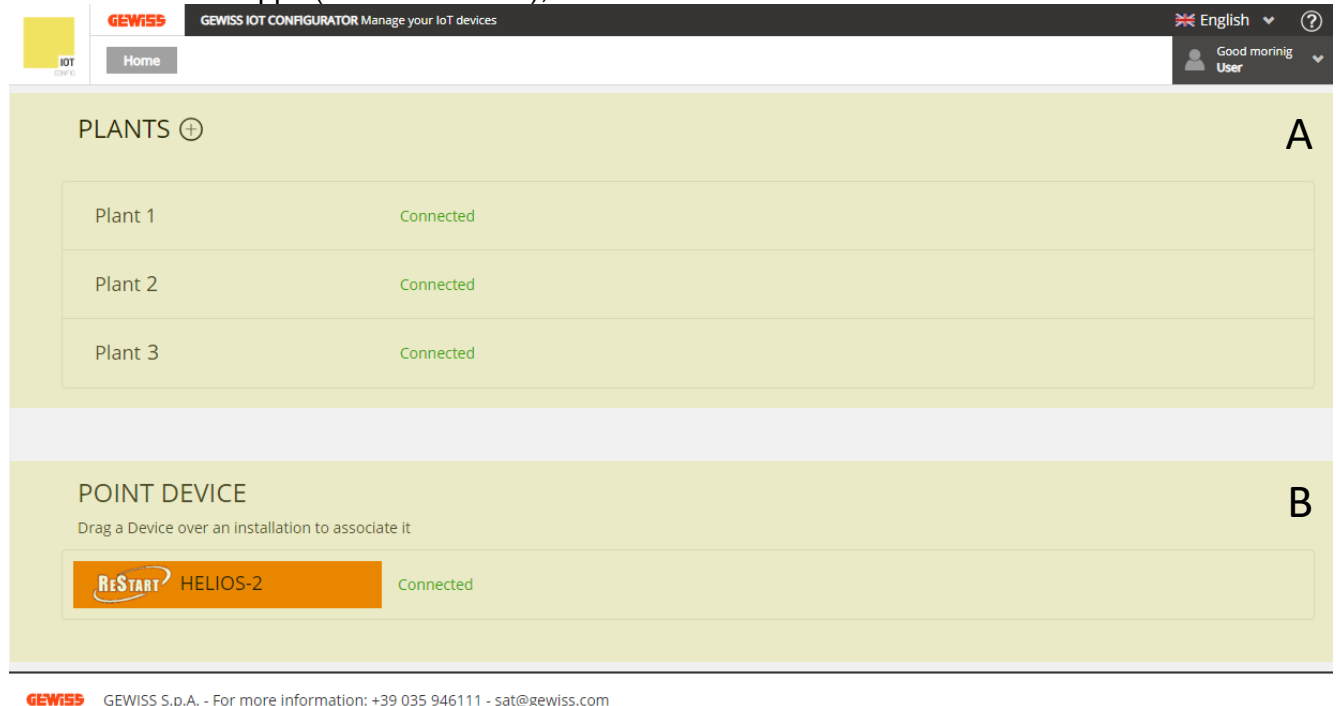
The system will send an automatic e-mail to the address given, with a link for completing the registration.

After clicking on the link, your registration is confirmed and you will be automatically redirected to the IoT Configurator access page, where you can enter your credentials and go to the portal.
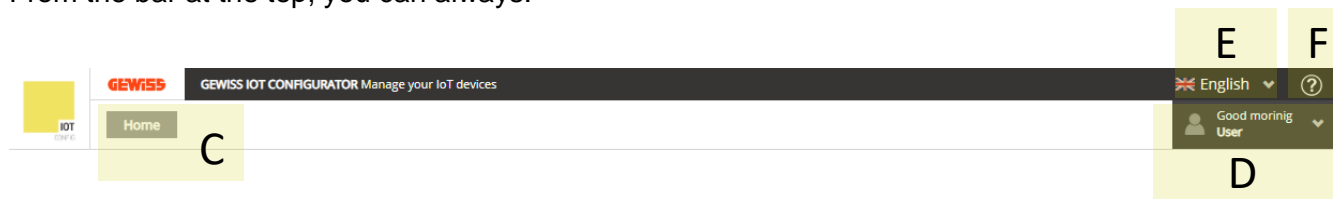
IoT Configurator homepage

Once you have accessed the IoT Configurator, you will see the homepage showing:
A. a list of the systems associated with your account, indicating the status of the IoT devices in it
B. the IoT devices not yet associated with a system but associated with your account via the use of the dedicated apps (POINT DEVICE), and their relative connection status

From the bar at the top, you can always:

C. return to the homepage
D. access the "Modify account" and user disconnection section
E. change the presentation language
F. see the release version of the IoT Configurator and the and the configuration section of the Smart Gateway App

Creating a new system

From the homepage, you can create a new system just by pressing the "+" icon next to the item "Systems".

In order to create a new system, you will be asked to enter the ID of your GEWISS IoT device; the identification code is shown on the label on the device itself (see the Smart Gateway example below)

After you have entered the code, a check is automatically made to ensure the device in question isn't associated with another system:

a. if the outcome is OK, the system is created and the user is associated with "Installer" rights
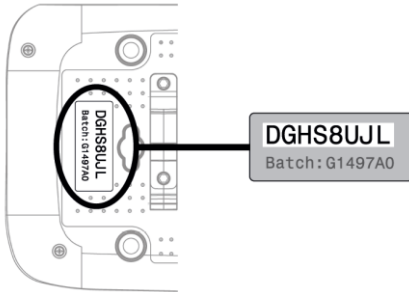b. if the Gateway is already associated with another system, you are asked which role you want to adopt (from the list of those available) and the association is then made. If no role is available, it will not be possible to join the system

When the procedure has been completed, the name of the new system is added to the list.

Associating an IoT device with a system

To take an IoT device already associated with your account (e.g. via the use of a dedicated app) and associate it with an existing system, just select it and drag it onto the name of the required system.

Once you have made the association, the device will no longer be visible in the POINT DEVICE list; it will be shown in the list of devices of the system that it's been added to.

Chorus
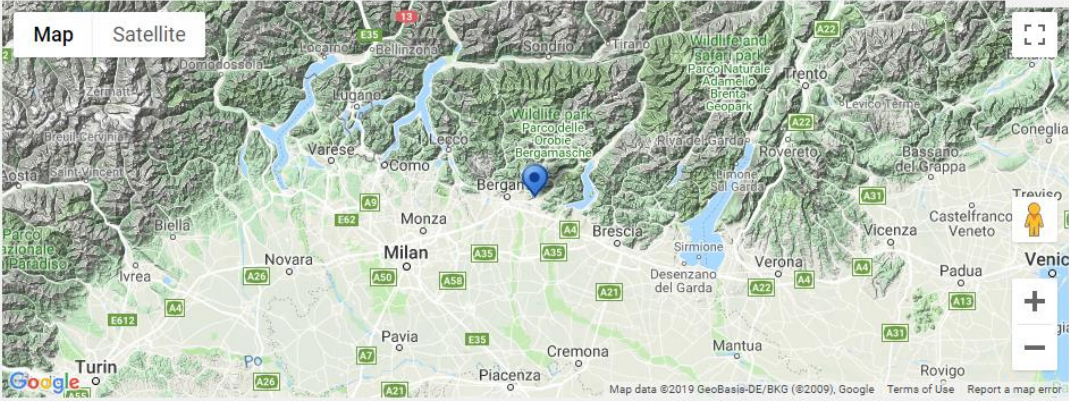
GEWISS IOT CONFIGURATOR Manage your IoT devices

English ❓

Home   Demo ▾

Good morinig
User

DETICE 🗝 🖉

DETAILS 🗝 🖉

Creation time:	28 Feb 2018 12:04:56
Latitude:	45.691
Longitude:	9.818

🏠 SMART GATEWAY App 🗝 ☰   License expiration: FREE PERIOD   👤 *system administrator*

⬜ Demo   Connected   🗝 ⬇ ☰

🏠 RESTART App ☰   👤 *system administrator*

🔲 Helios-3   Connected   ⬇ ☰

SYSTEM MANAGEMENT

In the 'Systems' section of the IoT Configurator homepage, you can find a list of all the systems associated with the user's account, and the relative status of the devices. Click on the name of a system to see all the details.
Position the mouse on the name of a system to visualise other commands.

1. delete the system
2. rename the system
3. access the system details

System details

When you select a system from the homepage, a page of details is called up:

A. **Navigation bar**

The navigation bar shows the name of the selected system. You can change the system by pressing on the drop-down menu (without returning to the homepage).

B. **System details and geolocation**

This section of the page shows general information and the geolocation of the system.
At the time of creation, information about the latitude and longitude are obtained automatically from the operating system of the PC used; this information can be modified at any time by selecting the
⊘ icon underneath the map.

C. **Supervision app and the relative devices controlled**

Below the system name you can see all the apps and a list of all the IoT devices that are controlled. For each app, there's the name and (at the top right) the role associated with the user for that specific app.

Alongside the name, you'll find the following commands:

🔧 Configure: It allows to access the IoT Configurator section where you can configure the app and the functions of the devices in the system

The optional commands menu (activated via the ≣ push-button) contains:

⊕	Aggiungi Smart Device	It adds a new IoT device (not associated with another system/user) to the system. This command is not available for Smart Gateway App
⬅	Importa Configurazione	It allows to replicate the programming of the Smart Gateway App of another system associated to the user (with Smart Gateway present) or to import the configuration from file
➡	Esporta Configurazione	It allows to save the Smart Gateway App programming on file
🔑	Rinnova licenza	It allows to access the IoT Configurator page where you can renew the app user licence
🗑	Elimina tutti gli Smart Gateway	It allows to eliminate the Smart Gateway associated with the system and disassociates all users from the Gateway

Next to the name of an IoT device controlled by the app, there is the connection status and the following commands:
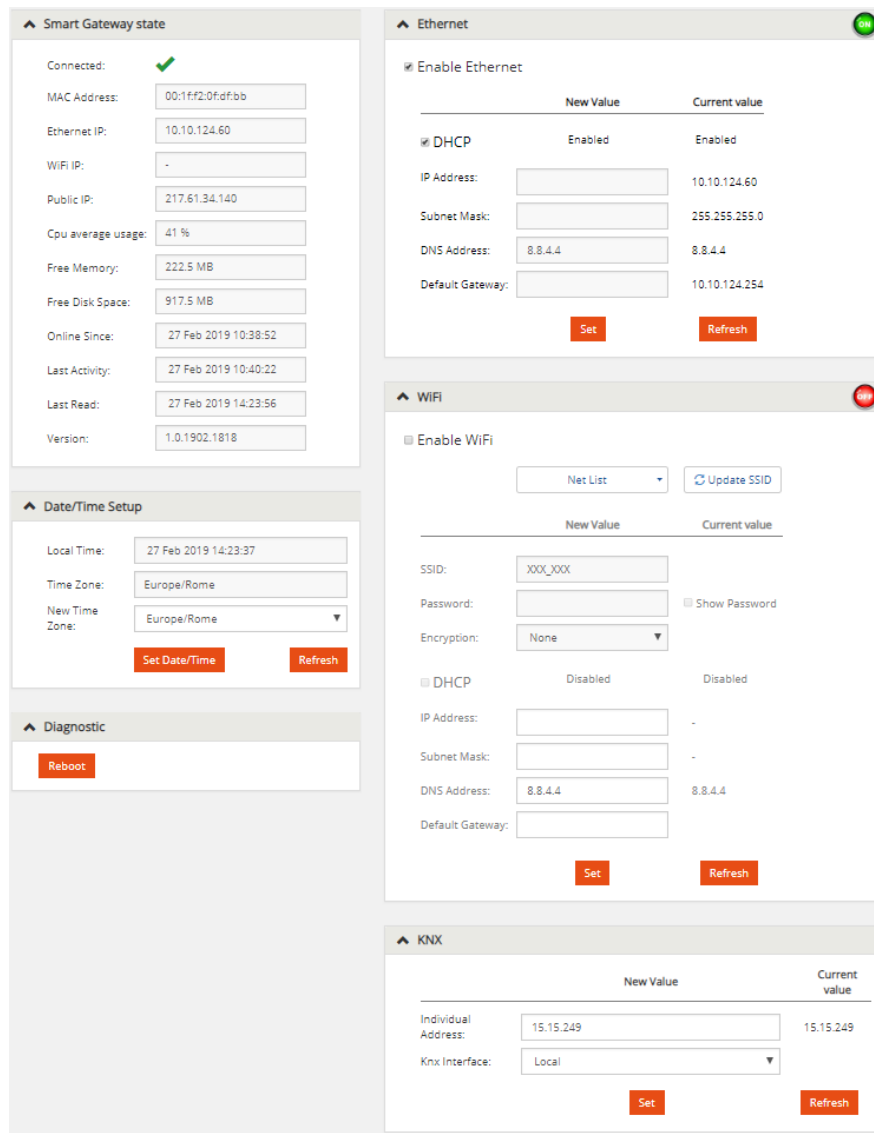
🔧	It allows to set device parameters: for accessing the page where you can configure the device operating parameters
⬇	It allows to update the device firmware: for updating the firmware version of the device (to launch the update, the device status must be "connected"). When a more up-to-date version of the firmware is available, the button is orange coloured and a message is displayed under the connection status of the device

The optional commands menu (activated via the ≣ push-button) contains:

AI	Rinomina	It allows to rename the device (i.e. change the name of the device)
↻	Sostituisci	It allows to replace a device (activates the procedure to replace a faulty device, transferring the configuration and parameters to the new one)
🗑	Elimina	It eliminates the Smart Gateway-system association and all the associations between user-Smart Gateways

Setting the Smart Gateway parameters

From the details page of a system, select the icon Ⓔ of a Smart Gateway to access the page where you can set the operating parameters of that Gateway.

- **Smart Gateway state**
  This section contains all the information about the connection status of the Smart Gateway and the firmware version used.

- **Date/Time setup**
  This section has all the information relating to the current date and time indicated by the Gateway; you can alter the time zone of the device.

- **Diagnostic**
  In this section it is possible to remotely restart the device.

- **KNX**
  In this section, you can select the method for communication between the Gateway and the KNX devices: local (Twisted Pair line connected to the KNX terminal) or via an external KNX/IP interface installed in the system.
  If you select "local" (default configuration), you can modify the physical KNX address of the Gateway (15.15.249 by default).

- **Ethernet**
  In this section, you can enable the Ethernet connection of the device and set the relative parameters. The Ethernet and wi-fi connections can be activated simultaneously; in this case, the Ethernet connection has the higher priority.

- **Wi-fi**
  In this section, you can enable the wi-fi connection of the device and set the relative parameters (wi-fi network to be connected to, and network parameters). The Ethernet and wi-fi connections can be activated simultaneously; in this case, the Ethernet connection has the higher priority.

Deleting a system

From the homepage, select the icon ⑩ related to a System to delete it from the user list.
The last user who performs the operation will be notified of the fact that the data stored in the cloud related to the system will be maintained for 25 months, unless it is expressly requested to cancel them instantly.

CONFIGURING THE FUNCTIONAL BLOCKS AND FUNCTIONS OF THE SMART GATEWAY

From the system details page, select the icon ✐ next to SMART GATEWAY App to access the page where you can configure the supervision and functions of the Smart Gateway.
The configuration is only possible if the PC that you are using is connected to the Internet; you cannot configure - or save the configuration - without an active connection.
During the configuration, the Smart Gateway doesn't need to be connected to the Internet; once it is connected, it will check if there is a more recent version of the project than the one already uploaded, and download it.
The configuration page appears like follows:
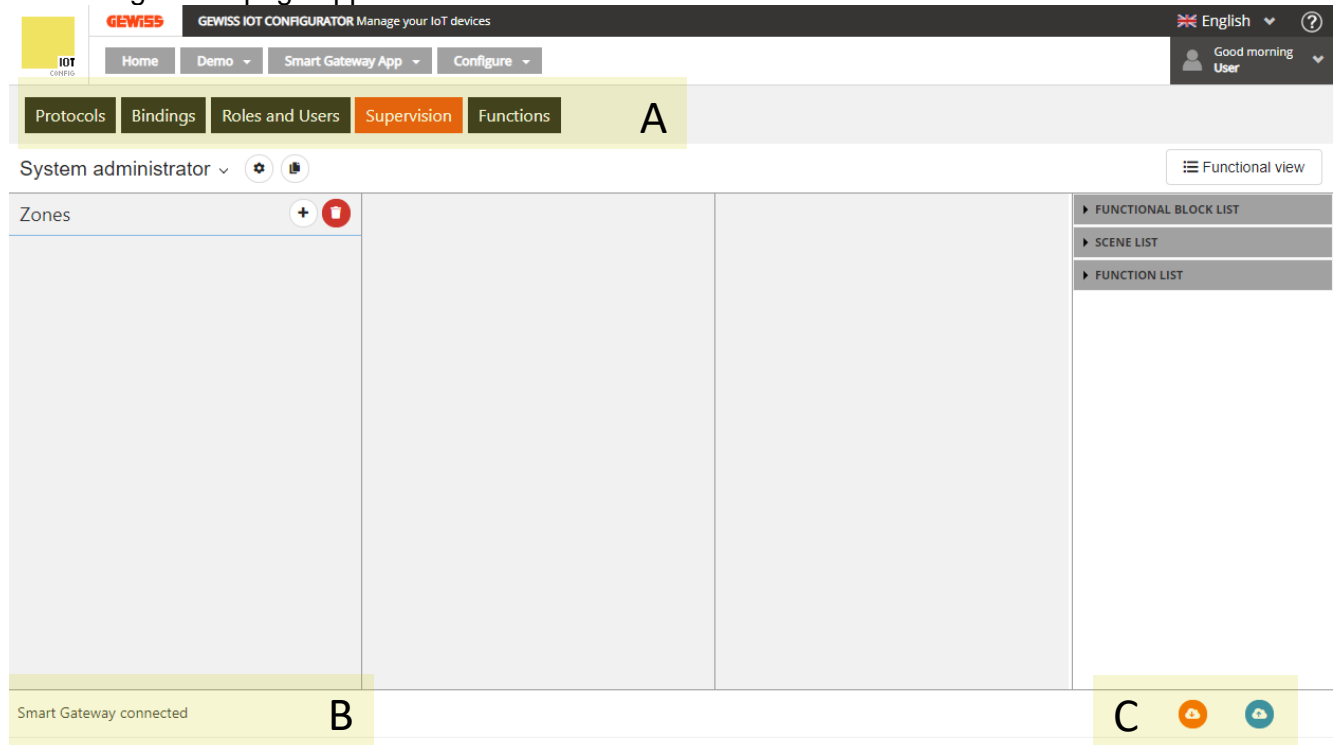
A. List of functions sub-divided into topic menus or sub-menus.
When you select a function, the central part of the page changes structure to allow the configuration.
B. Information relating to the Gateway and its connection status.
The connection status of the Gateway is shown; if you select the "ZIGBEE" menu, additional dedicated commands will appear (see Managing the devices and modelling the ZigBee functional blocks).
C. Send configuration / Read configuration.
These two push-buttons are used to save the configuration on the cloud or download the current configuration from the cloud.
The project is saved whenever a modification is made; the "temporary" project is saved on the cloud too, but in a dedicated area that can't be accessed by clients (Smart Gateway and app).

When the ☁ push-button is pressed, the file is moved from the temporary area to the area shared with clients (that, when they connect, can download it).

The ☁ push-button is used to take the current configuration and copy it in the temporary area (if it has been modified via the app). The push-button changes colour to show:

☁ the temporary configuration coincides with the shared one, but the Smart Gateway hasn't yet downloaded it

☁ the temporary configuration does NOT coincide with the shared one

☁ the temporary configuration coincides with the shared one, and the Smart Gateway has already downloaded it

NOTE: before beginning to modify a configuration on which you haven't worked for some time, YOU ARE ADVISED to read the current configuration to make sure the project is aligned.

Modelling the KNX devices/functional blocks

The KNX devices in the system must be programmed via the ETS tool (distributed by the KONNEX association). The Smart Gateway doesn't have a KNX database, but it does allow the ETS project to be imported in order to model the devices/functional blocks within the IoT Configurator.

After completing the programming, export the project in ".knxproj" format and then import it in the IoT Configurator; to import a KNX project, select **Protocols→KNX** and press **🖐 Import KNX project** icon.
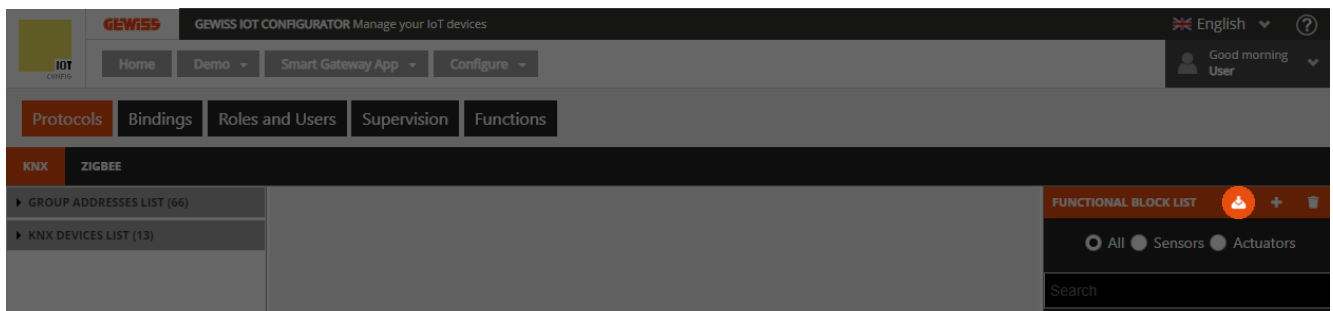
Select the file and then confirm its opening.

The conversion tool will start processing the KNX project. When this step has been completed, the left-hand column will show the group addresses in the ETS project.

The group addresses are arranged in two different ways:

- **LIST OF GROUP ADDRESSES**

A complete list of the addresses, in ascending order (visualisation at three levels). For each address, you can see the name, size and Data Point Type (if specified in ETS).
At the head of the list there's a text search field for filtering the contents: the search can be made on the basis of address, address name, size or Data Point Type.

- **LIST OF KNX DEVICES**

A tree structure showing:
i. the name of the KNX device (derived from the topology of the ETS project)
ii. the name of the communication object of the device, and the object function
iii. the group address associated with the communication object, along with the name, size and Data Point Type (if specified in ETS).
At the head of the list there's a text search field for filtering the contents: the search can be made on the basis of KNX device name, object name and function, name, size or Data Point Type group address.

After importing the group addresses from the KNX project, you must create the functional blocks that model the KNX devices. This process allows you to construct the supervision and, if necessary, create the application functions with devices that communicate via other protocols such as ZigBee. Bearing this in mind, it may be useful to model the sensor functional block that acts as a KNX contact interface so it can use the Gateway to command ZigBee actuators.

To create a new functional block, press the "**+**" icon in the right-hand column ("LIST OF FUNCTIONAL BLOCKS").
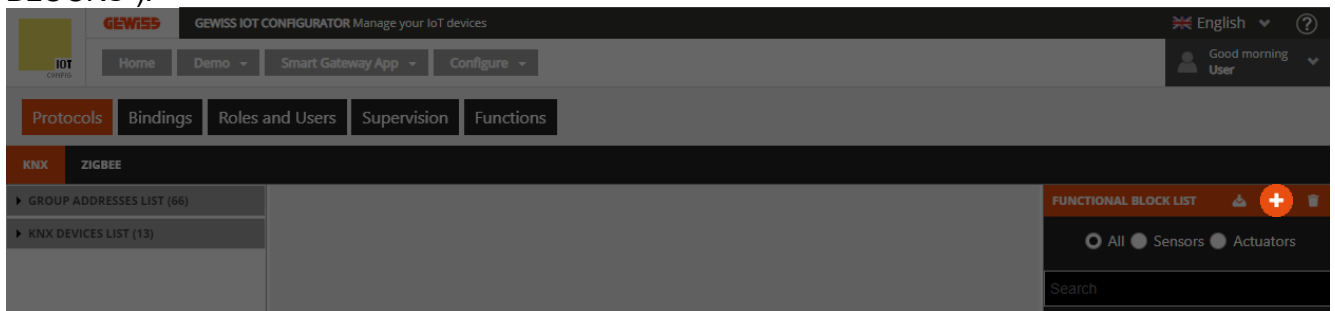
The pop-up menu will show the text field for the name you want to assign to the functional block and, underneath, a list of all the functional block models in the system. You can filter the list by selecting one of the two push-buttons - "Sensors" or "Actuators".
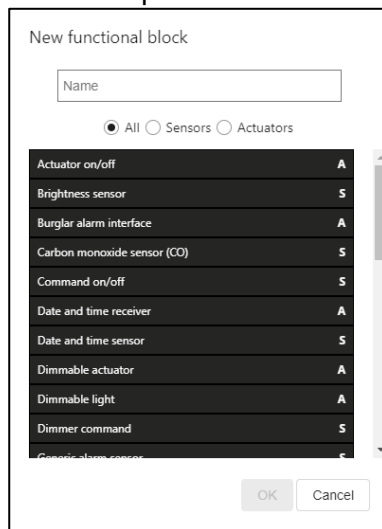
Apart from the name of the functional block model, the right-hand part also shows the category: Actuator (A) or Sensor (S).

After selecting the functional block and confirming your choice, the central part of the page will show the functional block model.
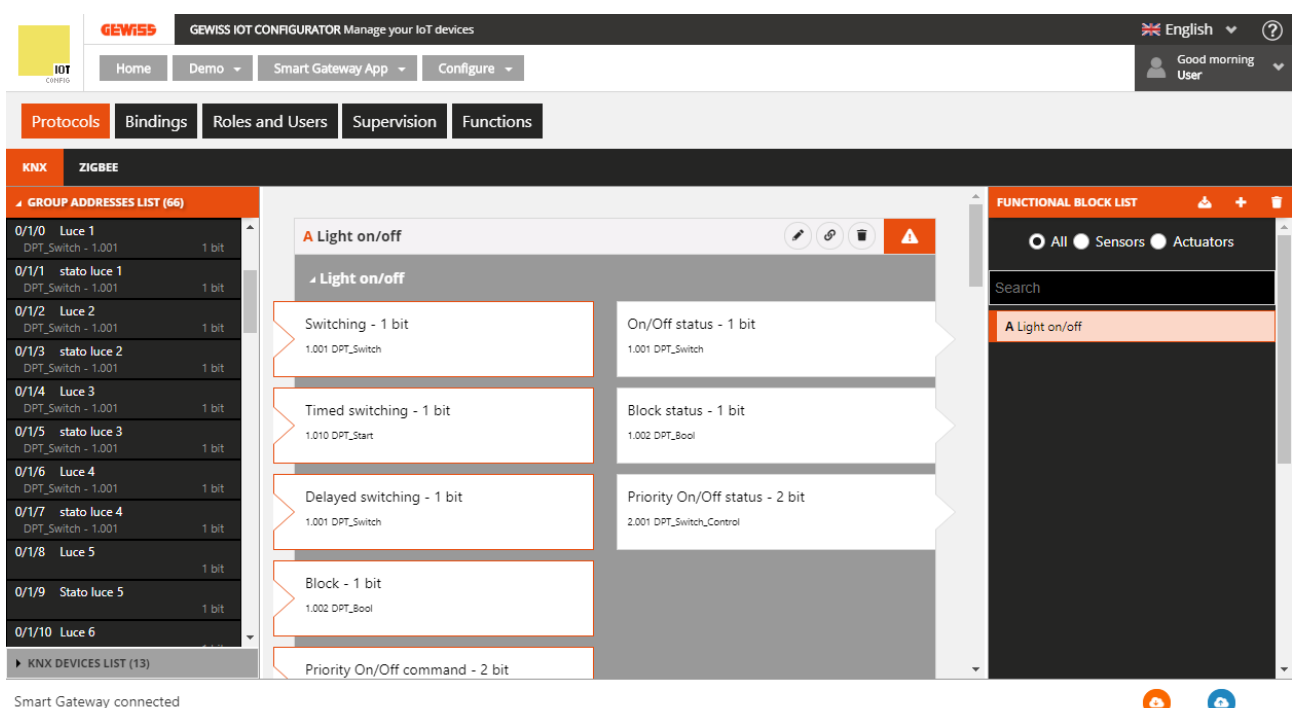
The category is indicated on the left-hand side of the functional block name. On the right, there is:

✏ **Rename** push-button for modifying the name of the functional block

🔗 **View connections** push-button for displaying all the pages where the functional block is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the functional block

⚠ ✅ Icon indicating the validity status of the functional block: orange if the functional block is incomplete and can't therefore be used in the project, green if the functional block is complete and can be used. If the functional block is incomplete, click on the orange icon to see the type of error.

In the area below, you will see the communication objects envisaged for that type of functional block: the input communication objects on the left, the output ones on the right.
Objects with an orange border are compulsory in order to validate the functional block; some are alternatives of others but, once an object is connected, the alternative ones are automatically disabled.

To associate a group address with an object, just select the address from the list (left-hand column) and drag it onto the required object. While the address is being dragged, the system compares the size of the group address with that of the functional block objects, temporarily disabling any that are incompatible with the address selected.
The object that is about to be associated with the address is highlighted in green. When the dragging operation is complete, the address is shown next to the name of the object.
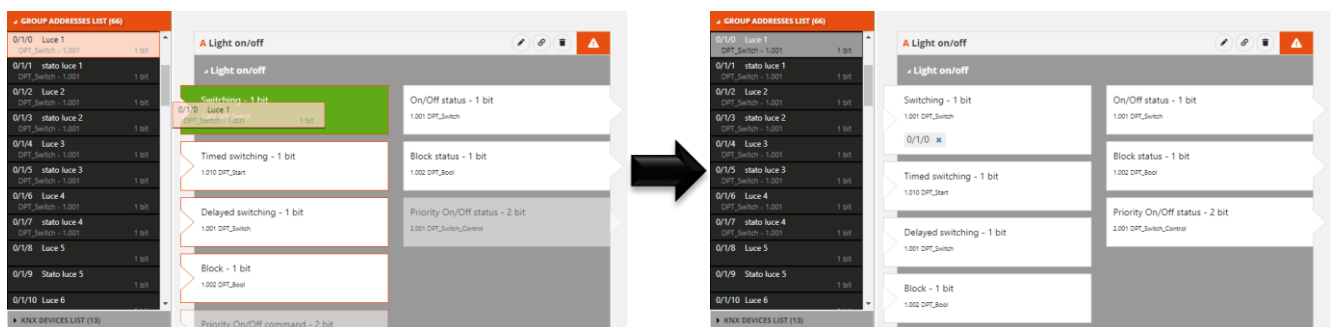
In the LIST OF GROUP ADDRESSES / LIST OF KNX DEVICES, the group addresses connected to the functional block are highlighted in grey.
If the objects that are compulsory for functional block validation are connected, then the functional block is valid and the ✅ icon appears.

ATTENTION: an incomplete functional block isn't saved on the cloud. This means that it will no longer be present when the Internet page is reloaded, or when the project is closed and reopened, even if the configuration has been sent to the area shared with clients (by pressing the ☁ push-button).

To delete a group address from an object, press the "x" `1/0/14 ✕` icon alongside.

All the functional blocks created are listed in alphabetical order in the LIST OF FUNCTIONAL BLOCKS column (on the right).
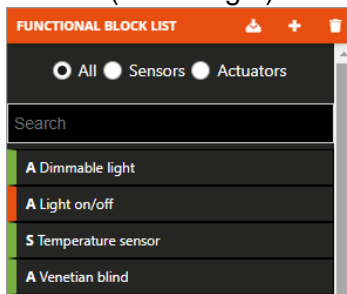
For each functional block, you can see:
- the name
- the category (A - Actuator or S - Sensor)
- the functional block validity status (orange - incomplete or green - complete).

At the head of the list there's a text search field and two option push-buttons "Sensors" or "Actuators" for filtering the contents.

To remove a functional block from the list, just activate the context menu on the functional block in question and select the "Delete" command. You can even eliminate all the functional blocks simultaneously by pressing the 🗑 icon in the right-hand column ("LIST OF FUNCTIONAL BLOCKS").
ATTENTION: a functional block can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the functional block won't be deleted.

Field scenes (pre-set)

In a pre-set scene, the objects or devices belonging to the scene are predefined in the configuration or initial start-up phase, and the commands are all implemented at the same time.
When the **Functions → SCENES** menu is selected, the list of SCENES appears (on the right) along with the list of elements that can be used in those scenes.
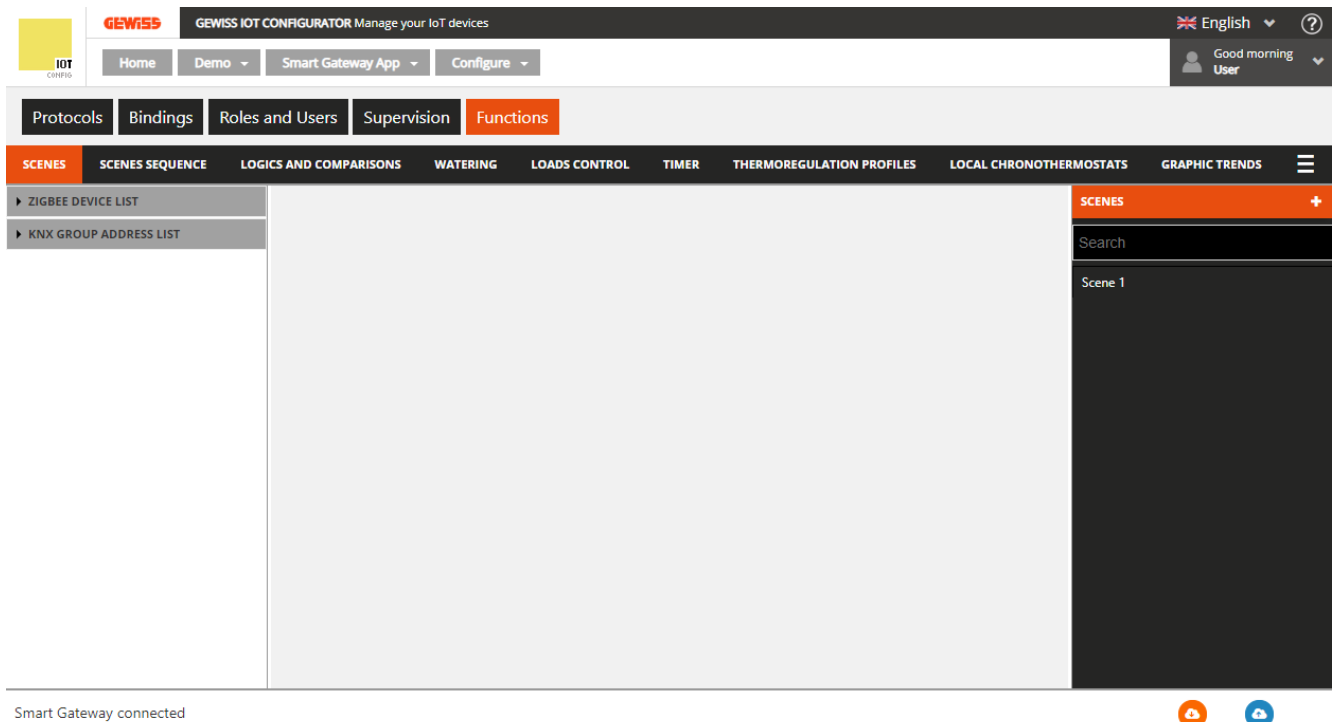
To create a new scene, press the "**+**" icon in the right-hand column ("SCENES").
Enter the name and ID (from 0 to 63) that you want to associate with the scene. Depending on the elements included, the ID will be used as:
- the number of the scene to be called up or learnt (for the KNX part)

- the ID of the receivers of the commands (for the ZigBee part); the scene ID is fixed at 1

Once the ID has been set, it cannot be modified.

After confirming your choice, the central part of the page will show:
- two columns - Sensors and Actuators (as for the Application functions) - relating to the ZigBee functional blocks
- a line at the bottom for the KNX group addresses

Next to the name of the scene you will see:

✏️ **Rename** push-button for modifying the name of the scene

🔗 **View connections** push-button for displaying all the pages where the scene is used (select the required page to gain direct access)

🗑️ **Delete** push-button for eliminating the scene

The left-hand column shows:

- **LIST OF ZIGBEE DEVICES**
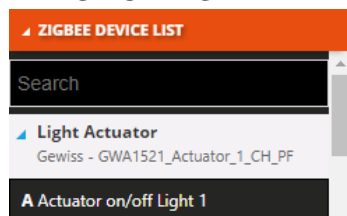
A tree structure showing:
  i. the name of the ZigBee device (taken from the "ZIGBEE" menu)
  ii. the device functional block, with name and category - Actuator (A) or Sensor (S).

If a functional block (sensor) is highlighted in grey, this means it has already been used in another scene or application function, and can no longer be used.

If a functional block is highlighted in blue, it has been inserted in the scene you are editing; when you bring the cursor onto a functional block shown in the central area, that functional block will be highlighted in green in the LIST OF ZIGBEE DEVICES.

- **LIST OF KNX GROUP ADDRESSES**
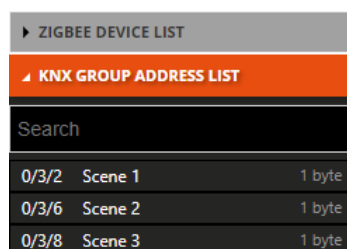
A complete list of the addresses, in ascending order (visualisation at three levels), of 1 byte dimension. For each address, you can see the name, size and Data Point Type (if specified in ETS).

At the head of the list there's a text search field for filtering the contents: the search can be made on the basis of address name or Data Point Type. The group addresses associated with the scene you are editing are highlighted in grey.

Select a ZigBee functional block and drag it into the central area in order to associate it with the scene you are editing; if it's an Actuator type functional block, a pop-up menu will appear so you can select:
- the cluster of the functional block involved in the scene
- the initial value of the attributes of the cluster selected
- the transition time - i.e. the time, in seconds, that the ZigBee device must take to switch from its current status to the one requested by the scene execution command; this is a parameter of the ZigBee protocol, managed directly by the device itself

The functional block will be visualised in the central part of the page, in the Sensors or Actuators column depending on its category.

A. the name of the device functional block (taken from the "ZIGBEE" menu)
B. the name of the functional block model
C. contextual functions:

- Modify the parameters of the actuator functional block (the same ones visualised when the functional block is inserted in the scene)

- View the details of the functional block (connection to the ZIGBEE menu with the functional block selected).

- Delete a functional block from the scene

D. the type of functional block

There's no limit to the number of ZigBee functional blocks that can be associated with a scene, apart from the physical memory of the ZigBee devices that have to memorise the programming.

When there is at least one ZigBee functional block in the scene, the **Save scene** push-button is visualised, allowing you to write the programming on the physical ZigBee devices:

the scene modifications have not been programmed on the devices; press the push-button to send the programming

the scene configuration has been transferred

To associate a KNX group address with a scene, just select the address from the list (left-hand column) and drag it onto the centre of the page. The address will then be visualised on the line at the bottom - **KNX group addresses**. In the LIST OF KNX GROUP ADDRESSES, the group addresses connected to the scene are highlighted in grey.

To delete a group address from a scene, press the "x" 1/0/14 × icon alongside.

A scene is valid if at least one ZigBee actuator functional block or a KNX group address is present.

All the scenes created are listed, in the order they were created, in the SCENES column (on the right).
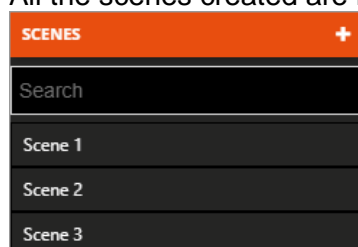
To remove a scene from the list, just activate the context menu and select the "Delete" command.
ATTENTION: a scene can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the scene won't be deleted.

To create a copy of a scene, just activate the context menu and select the "Copy" command. You will be asked to enter the name and ID of the new scene in the pop-up that appears. When the settings have been confirmed, a new scene will be created with the same functional blocks/communication objects used in the original one.

Creating roles and a role/user association

In the structure of the Gewiss application, each specific "role" can have different rights of access to the supervision views - i.e. the possibility to access zones, environments, functions, specific control elements (e.g. an object command).
In the Roles and Users section, you can create different roles within the installation and associate them with users. A user can have one role only within a single system.
The system envisages two roles that are always present and can't be deleted: Installer and System Administrator.
The first user associated with the Smart Gateway adopts the role of Installer (see Creating a new system); the other users are associated during the configuration phase.

The rights associated with the roles are as follows:

INSTALLER
This role is present by default (it doesn't need to be created and can't be deleted).
This role can only be covered by one user on each system.
The role must always be covered by a user; it's impossible to have a system without an Installer.
The Installer role gives the following rights:

1. View the system details
2. Update the software of the IoT devices
3. Enable the system: create, view, modify and delete the system configuration (Supervision and Smart Gateway functions)
4. Create, view, modify and delete user roles and specify a list of users who will cover that role for that system
5. Assign a user with the System Administrator role (one user only); this user cannot be one of the specific physical users of each user role, and must not already be associated with the Smart Gateway in another role
6. Create, view, modify and delete the supervision of each role
7. Enable/Disable the possibility for new users to associate themselves with a Smart Gateway
8. Enable/Disable the possibility of remote intervention on the system configuration by Gewiss Service
9. Change the roles of the physical users already associated with the system
10. Replace or delete the user associated with the System Administrator role
11. Change the user associated with the Installer role for a new user already registered on the GEWISS IoT Configurator; after making the replacement, the Installer will lose all configuration rights and will be redirected to the list of systems
12. Disassociate the physical users associated with the Smart Gateway
13. Disassociate himself from the Smart Gateway (before doing this, he must indicate another user for the role of Installer)

SYSTEM ADMINISTRATOR
This role is present by default (it doesn't need to be created and can't be deleted).
This role must be defined because the system administration rights must be assigned to a user other than the Installer at the end of the start-up. This role is usually adopted by the system owner.
The System Administrator role gives the following rights:

1. View the system details

2. Modify the system configuration: create, view, modify and delete the system configuration (Supervision and Smart Gateway functions)
3. Create, view, modify and delete user roles and specify a list of users who will cover that role for that system
4. Assign/Remove system modification rights to/from the Installer (this can also be done via the app): this operation takes away the Installer's right to access the system supervision function and all the other rights of the role, apart from the possibility to view the system details.
5. Change the user associated with the Installer role
6. Enable/Disable the possibility for new users to associate themselves with a Smart Gateway
7. Enable/Disable the possibility of remote intervention on the system configuration by Gewiss Service
8. Change the roles of the physical users already associated with the system
9. Disassociate the physical users associated with the Smart Gateway
10. Disassociate himself from the Smart Gateway

USER / END USER:
No limits to the User roles, that can be created by the Installer or System Administrator.
Users with a "User" role have no access to the App configuration section.
User roles can be utilised to differentiate supervision, in terms of visualisation/access to environments and command elements and the right (via the app) to create functions.
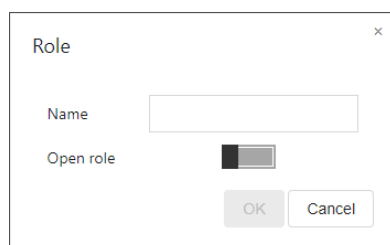
Once the "Roles and Users" menu has been selected, the LIST OF USERS is displayed on the right, while in the left are displayed the list of roles available and the access permissions to the plant.
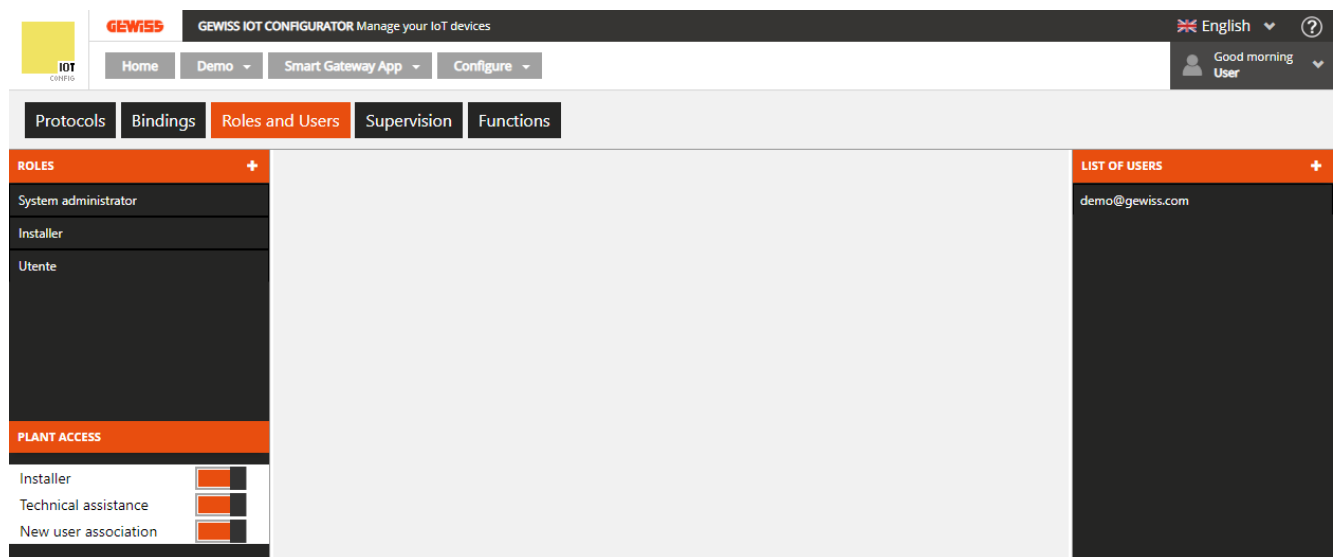
To create a new User Role, press the "**+**" icon in the right-hand column ("ROLES").

Enter the name you want to associate with the role, and specify whether the role is open or closed:
- open
a role that can be covered by any user once the Smart Gateway ID code has been entered (as long as the new user association flag is enabled)

- closed
a role that can only be covered by users specified by the Installer or System Administrator on this page

Examples:

case a) Open "Child" role and no user specified
All the new physical users that associate themselves with the Smart Gateway can assume the "Child" role.

case b) Open "Child" role and 2 physical users - "Giovanni@demo.com" and "Fabio@demo.com" specified
All the new physical users that associate themselves with the Smart Gateway can assume the "Child" role. The users "Giovanni@demo.com" and "Fabio@demo.com" will automatically assume the "Child" role for that specific system once they have accessed the GEWISS IoT Configurator.

case c) Exclusive "Child" role and 2 physical users - "Giovanni@demo.com" and "Fabio@demo.com" specified
Apart from the physical users "Giovanni@demo.com" and "Fabio@demo.com", no other new users who associate themselves with the Smart Gateway can assume the role of "Child".
The physical users "Giovanni@demo.com" and "Fabio@demo.com" will automatically assume the "Child" role once they have accessed the GEWISS IoT Configurator.
To add another user ("Maria@demo.com") to the "Child" role, the Installer or System Administrator must associate the user with that specific role (indicating the user "Maria@demo.com" for the role of "Child").

case d) Exclusive "Child" role and no user specified
No new users who associate themselves with the Smart Gateway can assume the role of "Child".
To associate a user with the role of "Child", the Installer or System Administrator must associate that user with that specific role (indicating the user for that role).

Once the setting has been confirmed, the new User role will be added to the Roles list (left-hand column).
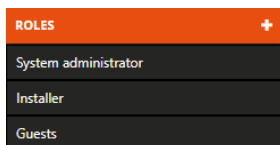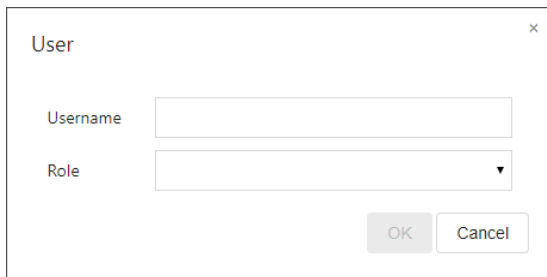
In the bottom of the left column, are present the parameters to enable/disable the access permissions to the system of:
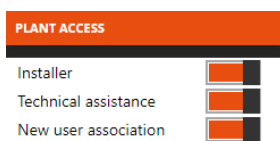
- **Installer** → It enables the installer to access the system (parameter visible only if the user has the role of system administrator). If disabled, the user who covers this role can no longer access the supervision and configuration of the system while remaining the installer
- **Technical assistance** → It enables the GEWISS technical assistance to access to the system
- **New users association** → It enables the association of new users to the system. If disabled, no user can be added to the system even if open roles have been foreseen
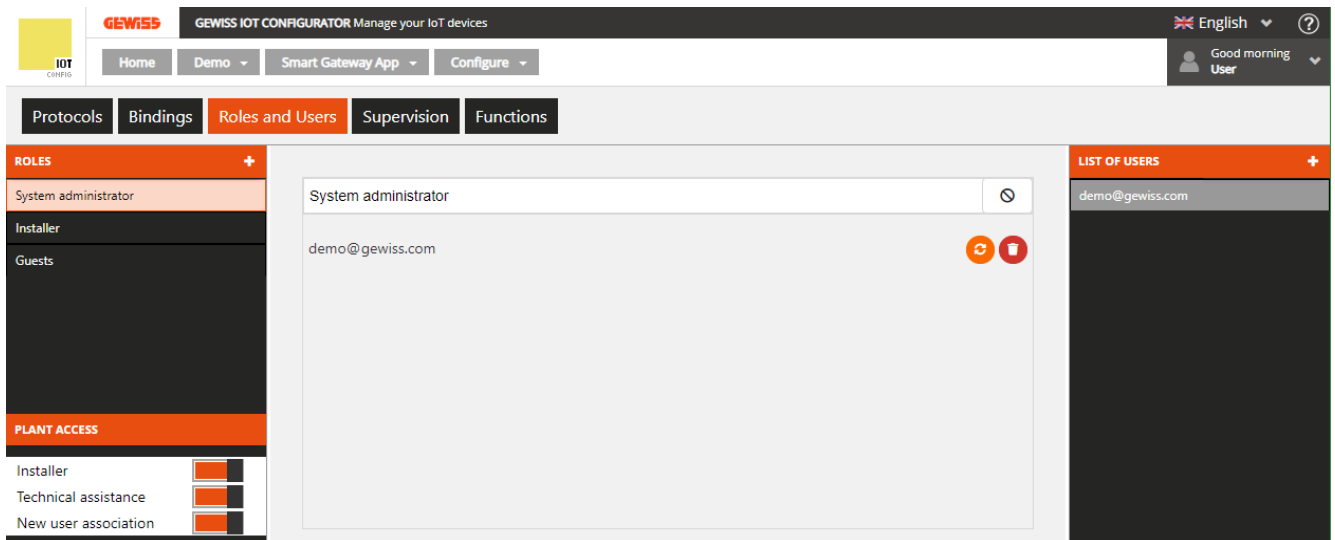
To create a new User to be associated with one of the associated roles, press the "**+**" icon in the right-hand column ("LIST OF USERS").

Enter the User name (valid e-mail address) and select the role you want to associate the name with.
The new user will be informed of the association by means of an automatic e-mail.
The LIST OF USERS shows all the users associated with the system.
In the LIST OF USERS (right-hand column), all the users associated with the role you are editing are highlighted in grey.

Once you have created a new role or selected one from the "ROLES" list (left-hand column), the central part of the page will show all the information relating to that role.

Next to the name of the role, you can see:

✏ **Modify** push-button for changing the name of the role and its type (open or closed); this push-button isn't available if the role selected is Installer or System Administrator

🚫 Icon showing that the role is closed
⭕ Icon showing that the role is open

Below the name of the role, there is a list of all the associated users. Next to each name on the list, you can see:

🔄 **Replace user** push-button for changing the user associated with the role; this push-button is only available if the role selected is Installer or System Administrator

🗑 **Remove user** push-button for deleting the user from that role and disassociating him from the system; this push-button isn't available if the role selected is Installer

A user's role can be modified by simply selecting the user name from the LIST OF USERS and dragging it directly into the list of users associated with the role you are editing; this operation automatically changes the role for that user.

In the LIST OF USERS (right-hand column), all the users associated with the role you are editing are highlighted in grey.

Scene sequence

A scene sequence can group together a series of commands to the devices in the field, with the possibility to delay their implementation; the Smart Gateway manages the implementation according to the settings configured.
After selecting the "Functions" menu on the first line and the "SCENE SEQUENCE" menu on the second, you will see the SCENE SEQUENCE column (on the right) and a list of the elements that can be used in the scenes in the column on the left.
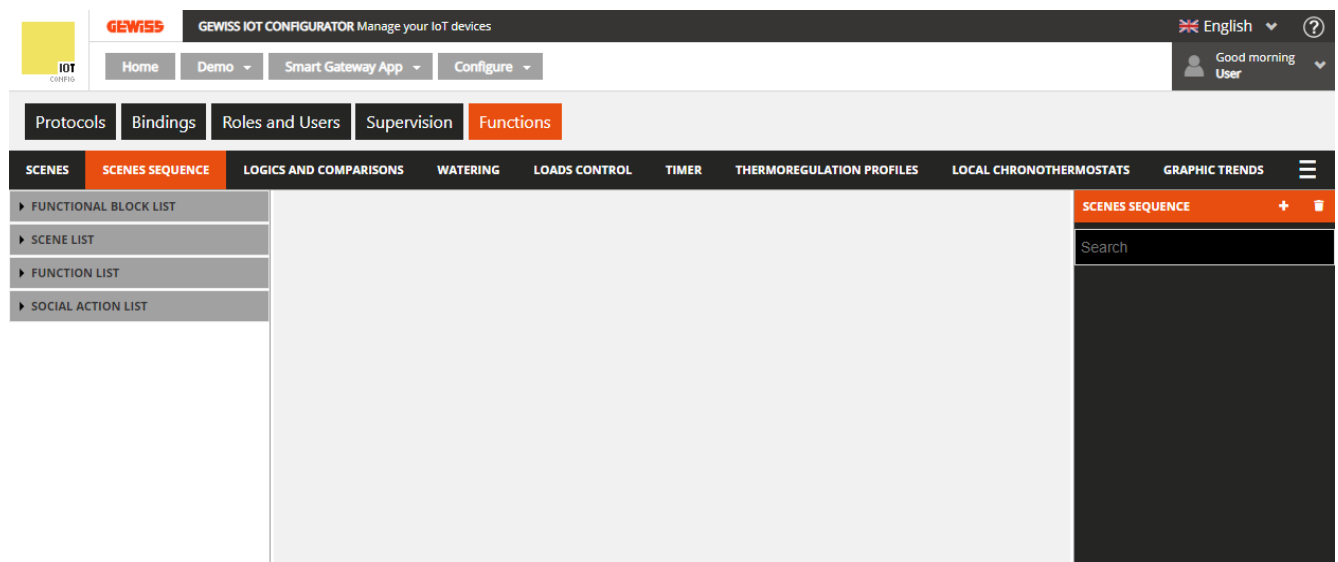
To create a new scene sequence, press the "**+**" icon in the right-hand column ("SCENE SEQUENCE"). Enter the name you want to associate with the scene, then confirm.
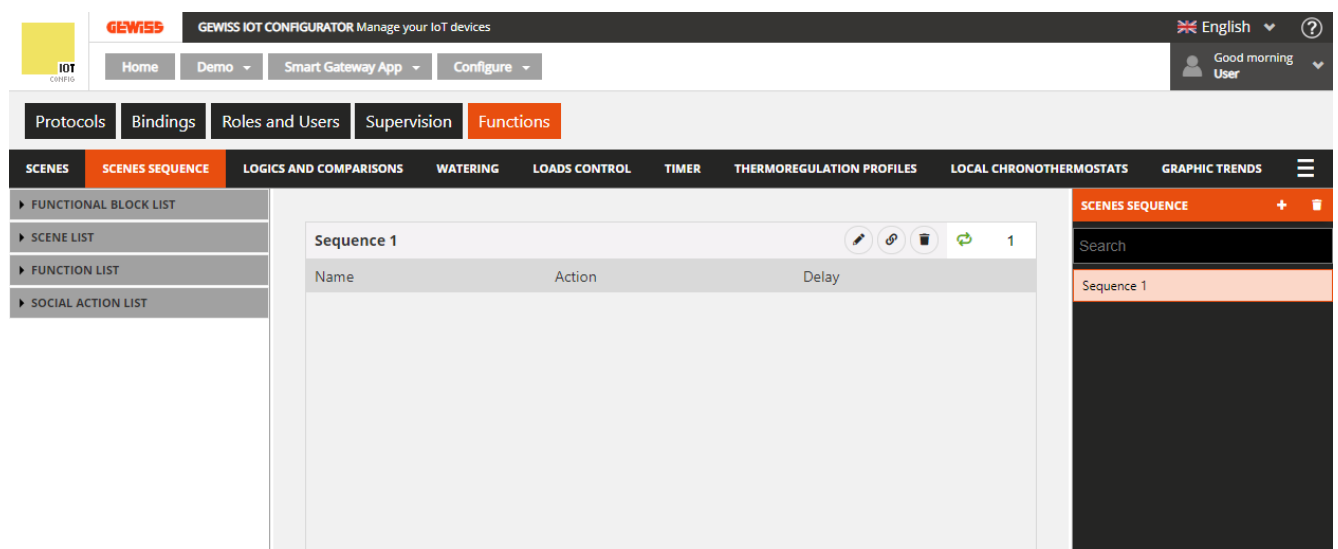
The central part of the page will show a list with all the actions (commands) that will be implemented when the scene sequence is activated.

Next to the name of the scene sequence you will see:

✎ **Rename** push-button for modifying the name of the scene

🔗 **View connections** push-button for displaying all the pages where the scene is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the scene

🔁 1 Numerical field for setting the number of repeats of the sequence of commands in the scene, once it has been activated. Possible values: from 1 to 250

The left-hand column shows:

- **LIST OF FUNCTIONAL BLOCKS**

◢ FUNCTIONAL BLOCK LIST
Search
◢ KNX FUNCTIONAL BLOCKS
A Dimmable light
◢ **Light Actuator**
Gewiss - GWA1521_Actuator_1_CH_PF
A Actuator on/off Light 1
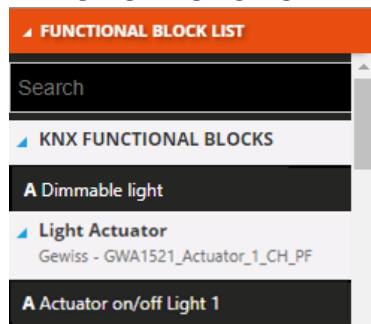
A complete list of the Actuator type functional blocks.
The ZigBee functional blocks are shown with a tree structure:
i. the name of the ZigBee device (taken from the "ZIGBEE" menu)
ii. the device functional block, with name and category ("A" = Actuator)
The KNX functional blocks are grouped in the KNX FUNCTIONAL BLOCKS sub-set and are represented with their name (taken from the "KNX" menu) and category ("A" = Actuator).
At the head of the list there's a text search field for filtering the contents.
The functional blocks already used in the scene sequence you are editing are highlighted in grey.

- **LIST OF SCENES**

▶ FUNCTIONAL BLOCK LIST
◢ SCENE LIST
Search
Scene 1
Scene 2
Scene 3
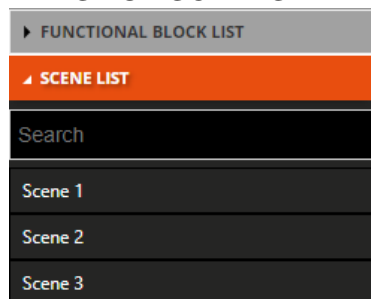
A complete list of the (pre-set) field scenes.
At the head of the list there's a text search field for filtering the contents.
The scenes already used in the scene sequence you are editing are highlighted in grey.

- **LIST OF FUNCTIONS**

▶ FUNCTIONAL BLOCK LIST
▶ SCENE LIST
◢ FUNCTION LIST
Search
Logic 1
Sequence 1
Timer 1

A complete list of the elements in the Functions category, with the exception of the preset scenes (separate list) and graphic trends (not managed).
At the head of the list there's a text search field for filtering the contents.
The elements already used in the scene sequence you are editing are highlighted in grey.

Select a functional block, scene or functions element and drag it into the central area in order to add it to the sequence you are editing; release the object in the list of actions, in the required execution position. At the end of this operation, a pop-up menu will appear where you can select:
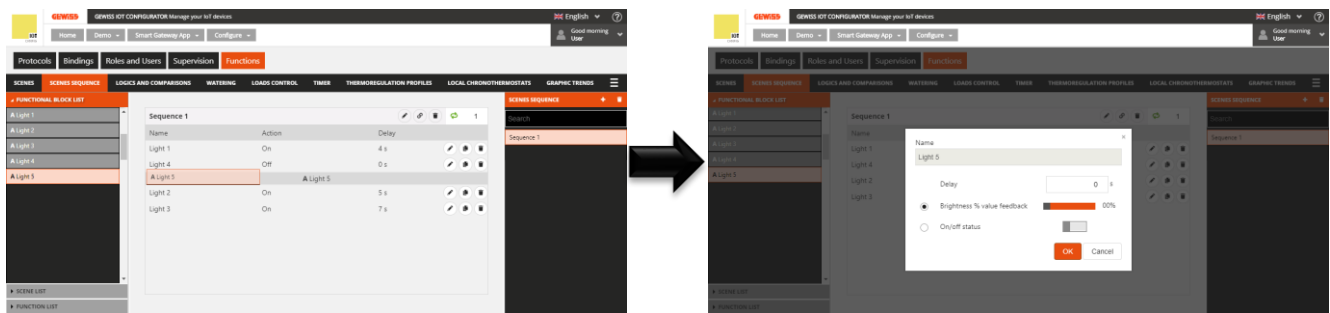
- the command implementation delay (in relation to the previous action in the list)
- the command you want to send (if several command types are available, you must first of all (before setting the value) select the command by ticking the relative checkbox).

After confirming the settings, the command will be visualised in the central part of the page, in the list of scene actions, in the position defined when you dragged it previously.

A. name of the functional block/scene/function to be managed via the command
B. command (action) to be implemented
C. implementation delay in relation to the previous action
D. contextual functions

✎   Modify the object parameters (the same ones visualised when the object was being inserted in the sequence); this function is called up by double clicking on the row that represents the object

📋   Create a copy of the action

🗑   Delete from the sequence

To move an action that has already been created, just select it and drag it to the required position.

There are no limits to the number of actions that can be associated with a scene sequence.

All the scene sequences created are listed, in the order they were created, in the SCENE SEQUENCE column (on the right).

To remove a scene sequence from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the scene sequences simultaneously by pressing the 🗑 icon in the right-hand column ("SCENE SEQUENCES").
ATTENTION: a scene can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the scene won't be deleted.
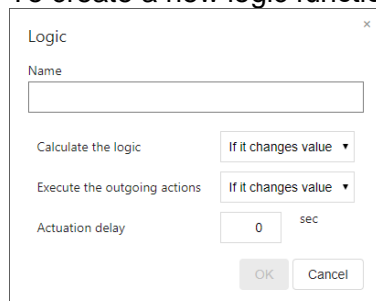
Logic functions and Comparisons

Through the logic functions it is possible to create conditioned actuations, which depend on the status of the various inputs or the result of other logic operations; for example, it is possible to connect the lights so that they are not both on, or raise the roller shutters if the wind becomes too strong. Actions can be associated on the basis of whether the result of the function is TRUE or FALSE.

The Smart Gateway manages the implementation of the actions according to the settings configured.

After selecting the "Functions" menu on the first line, and the "LOGIC FUNCTIONS AND COMPARISON" menu on the second, you will see the LOGICS column (on the right) and a list of the elements that can be used in the logics in the column on the left.
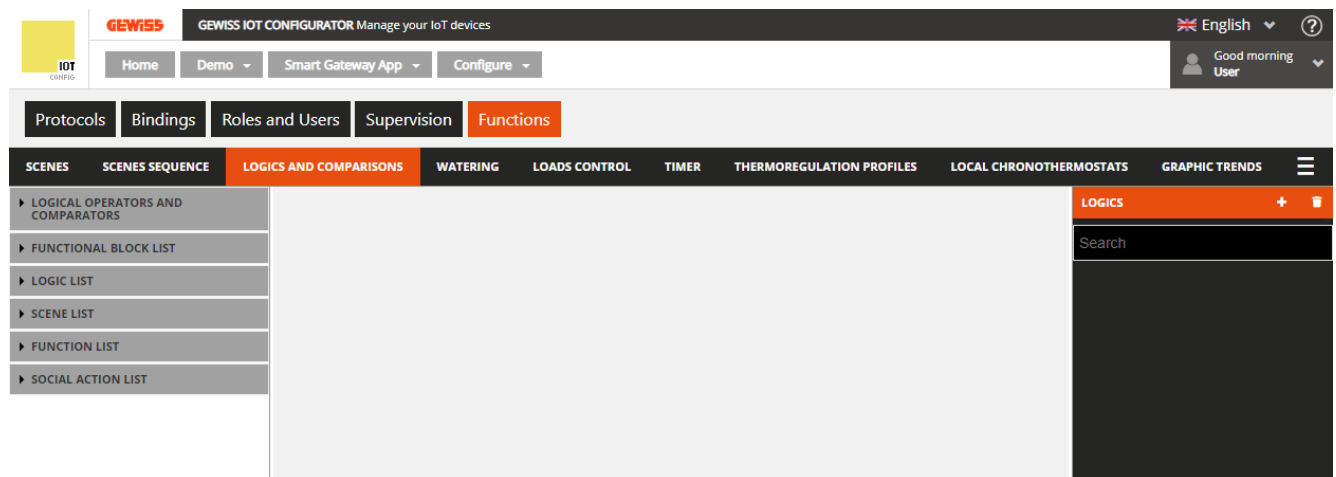
To create a new logic function, press the "**+**" icon in the right-hand column ("LOGICS").

In the pop-up menu that appears, enter the name and
- the logic calculation conditions (Calculation of the logic)
  a. If the value changes → the logic is calculated when the value changes at least one of the inputs
  b. Periodically → the logic is calculated at pre-fixed time intervals (with the duration defined in the "Calculation period" parameter)
  c. At each update → the logic is calculated each time an input value is received, even if the result does not change the value
- implementation conditions for the actions associated with the result of the logic function (Implement the actions on the output)
  a. If value changes → the actions are only implemented if the result of the logic changes
  b. With every update → the actions are implemented every time the logic is calculated, even if the result doesn't change
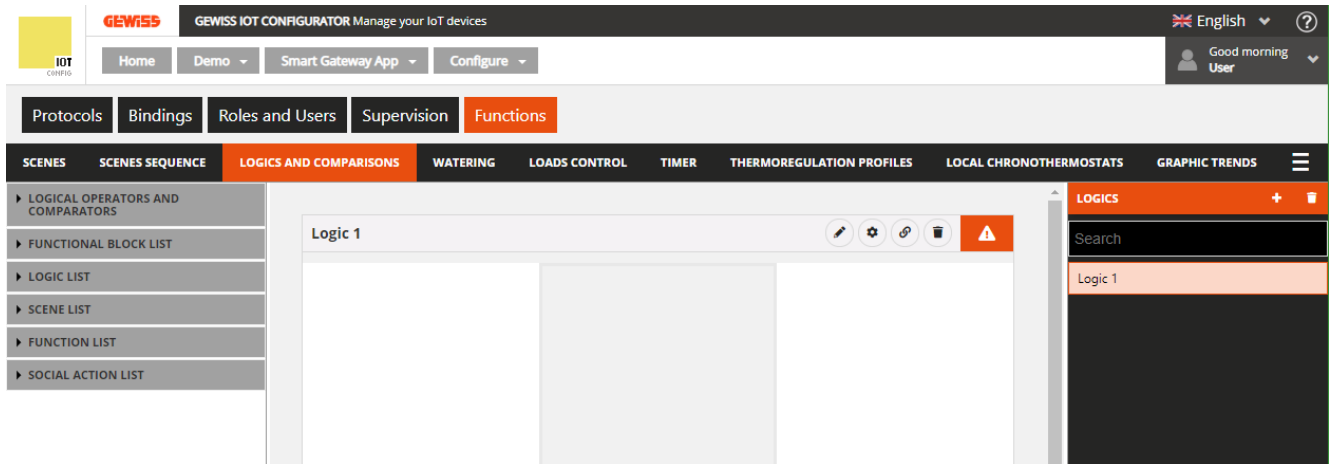
Once the choice has been confirmed, the central part of the page will show the area representing all the elements of the logic function and the relative connections.

Next to the name of the logic you will see:

⚙ **Modify** push-button for changing the name of the logic and the conditions for calculating and implementing the commands (the same ones visualised when the logic was created)

🔗 **View connections** push-button for displaying all the pages where the logic is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the logic

⚠ ✅ Icon indicating the validity status of the logic: orange if the logic is incomplete and can't therefore be used in the project, green if it's complete and can be used. If the logic is incomplete, click on the orange icon to see the type of error.

The left-hand column shows:

- **LOGIC OPERATORS AND COMPARATORS**

List of the operations:
- **Boolean operators (OR, AND, XOR)**, in light blue
  Needs at least two Boolean type inputs (0/1) connected
- **comparators (=, ≠, >, ≥, <, ≤)**, in green
  Needs two inputs connected
- **triggers 0→1|1→0 (switchover)**, in grey
  Needs one Boolean type input (0/1)
- **numerical input for comparisons** (numerical input), in yellow
  This isn't a logic operation; it's an input to all intents and purposes, to be used to compare sizes with predefined values (e.g. temperature > 25).

- **LIST OF FUNCTIONAL BLOCKS**
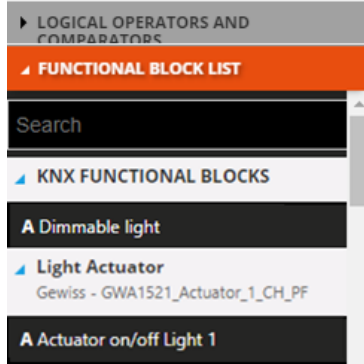
A complete list of the functional blocks.

The ZigBee functional blocks are shown with a tree structure:

i. the name of the ZigBee device (taken from the "ZIGBEE" menu)

ii. the device functional block, with name and category

The KNX functional blocks are grouped in the KNX FUNCTIONAL BLOCKS sub-set and are represented with their name (taken from the "KNX" menu) and category.

At the head of the list there's a text search field for filtering the contents.

The functional blocks can be used as either a logic input or output.

The functional blocks already used in the logic you are editing are highlighted in grey.

- **LIST OF LOGIC FUNCTIONS**
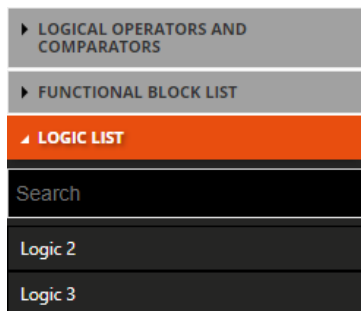
A complete list of the logic functions that have already been created (apart from the one you are editing).

At the head of the list there's a text search field for filtering the contents.

The logic functions can be used as either a logic input or output.

The logic functions already used within the logic you are editing are highlighted in grey.
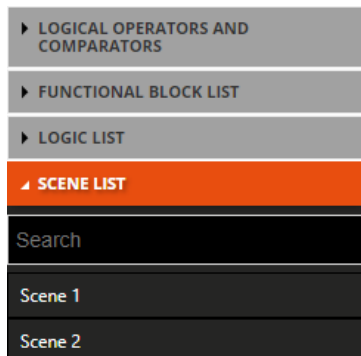
- **LIST OF SCENES**

A complete list of the (pre-set) field scenes.

At the head of the list there's a text search field for filtering the contents.

The scenes can only be used as a logic output.

The scenes already used in the scene sequence you are editing are highlighted in grey.
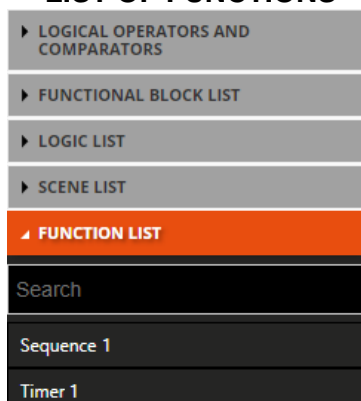
- **LIST OF FUNCTIONS**

A complete list of the elements of the Functions category, apart from Graphic Trends (not managed), Scenes and Logic Functions (separate list).

At the head of the list there's a text search field for filtering the contents.

The elements in this category can only be used as a logic output.

The elements already used in the scene sequence you are editing are highlighted in grey.

- **LIST OF SOCIAL ACTIONS**

▸ LOGICAL OPERATORS AND COMPARATORS
▸ FUNCTIONAL BLOCK LIST
▸ LOGIC LIST
▸ SCENE LIST
▸ FUNCTION LIST
◢ SOCIAL ACTION LIST
Push notification
E-mail

A complete list of the social actions available (Push notification, e-mail). The social actions can only be used as a logic output.

Select the Boolean logic operator or comparator and drag it into the central area of the page in order to add it to the logic you are editing. The operators are shown in the middle of the logic creation area.
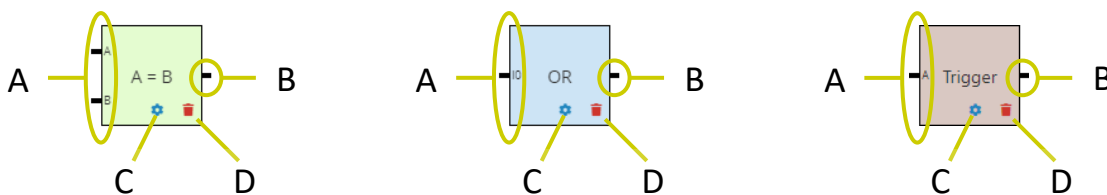
A. Connectors that identify the inputs of the logic operator; an input must be connected with each one of them.
   For the comparators, two non-Boolean inputs (A and B) must be connected.
   For the Boolean operators, at least two inputs (I0, I1); for each input connected, a new connector is created in order to link the next input (I2, I3, etc.).
   For the triggers (Switchover), just one Boolean input is needed.
B. Connector identifying the logic operator output.
   The following can be connected to the connector:
   - the input of another logic operator (only one), to create cascade operations
   - a functional block/logic/scene/function/social action on which actions can be implemented, without any limit to the number of elements connected
C. Access the logic operator settings (the parameters of the inputs and outputs that are connected). Alternatively, the settings can be activated by double clicking inside the graphic object that represents the operator.
D. Delete a logic operator

To insert an input, select the element (functional block, logic or numerical input) and drag it onto the left-hand part of the logic creation area; during this operation, a grey box will appear with the name of the element, to help you position the object in the creation area. When the element has been released, a pop-up menu appears where you can select the data item you want to use as the input.
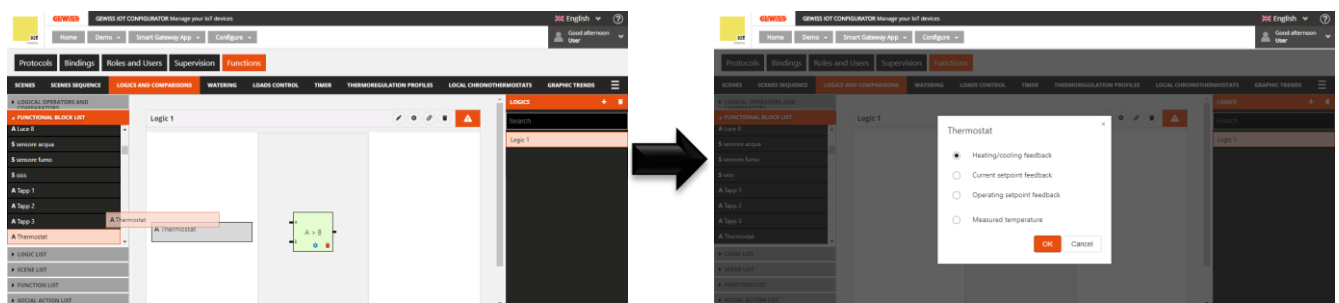
After confirming your choice, the element will be shown (name and data item selected) in the left-hand part of the logic creation area, in the position defined when you dragged it previously.
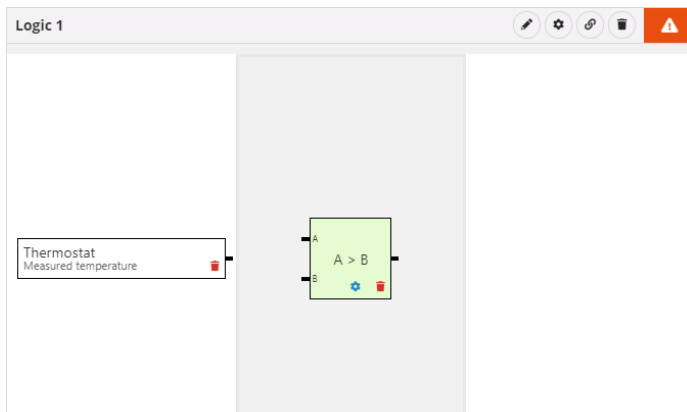
Press the 🗑 icon if you want to delete the element (even if it's already connected to an operator).

To make the connection between the logic operator and the input, select the connector that identifies the logic operator input and drag it onto the required element; during this operation, the border of the elements that are compatible with the logic operator will turn green. If an element is highlighted in green when the cursor moves onto it, this means the connection can be made.
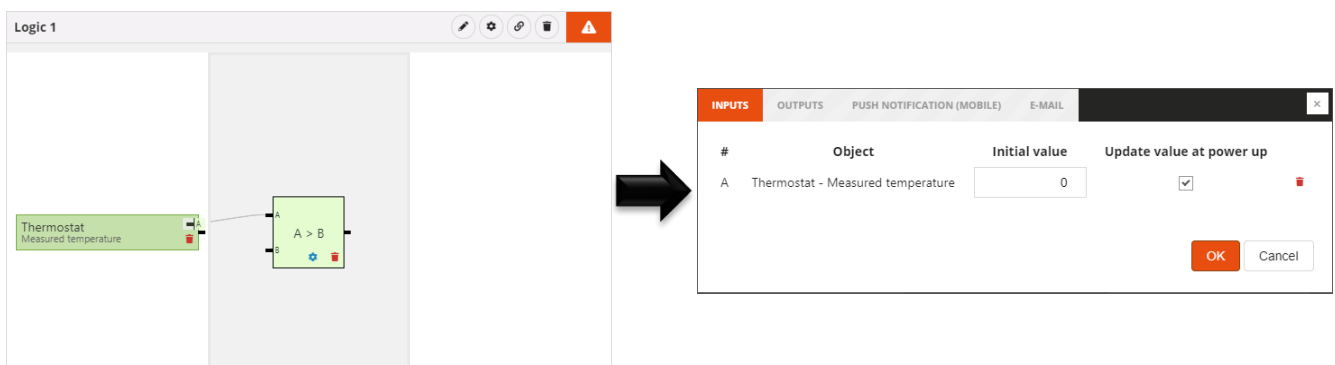
When the object is released, the logic operator settings relating to the inputs are shown:

→ name of the connector that the element is connected to
Object → name of the element and relative data item to be read
Not → enables the refusal of the value received from the field (FOR BOOLEAN OPERATORS ONLY); when the value of an input is refused, the connector is preceded by a white spot (see "I0" in the example below)
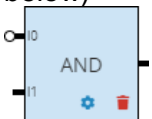
Initial value → Value which must be used for the purpose of the logic if the data from the field is not available; in the case of a "numerical input", the field is renamed **Value** and it's the value used to make the comparison (it can't be updated from the field).
Update data item at switch-on → enables the use of the data item read following a power supply reset; if this option is disabled, the "Initial value" will be used after the reset.
🗑 → deletes the element from the logic operator input (removes the connection between the two objects)

An input can only be connected to one logic operator. If you want to connect a single element to several logic operators, you must drag it into the logic creation area again and then connect it to the new operator (e.g. if you want to compare a temperature with two threshold values, minimum and maximum, the sensor functional block that measures the temperature must be inserted twice so the two objects can be connected to the two operators).

When the object is connected to the logic operator, you can remove the connection by means of the ⚔ icon.
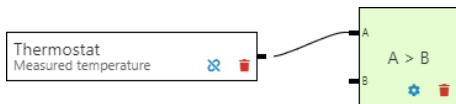
To insert an output, select the element (functional block, logic, scene or function) and drag it onto the right-hand part of the logic creation area; during this operation, a grey box will appear with the name of the element, to help you position the object in the creation area. When the element has been released, a pop-up menu appears where you can select the type of command you want to use as the action.
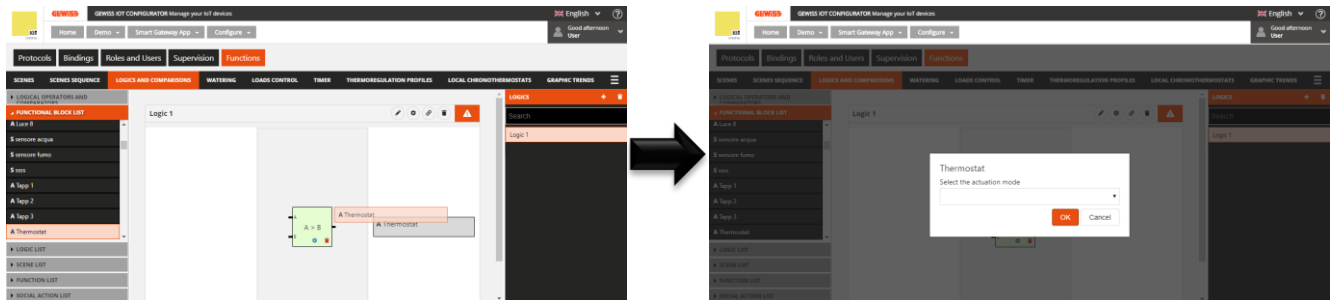
After confirming your choice, the element will be shown (name and command type selected) in the right-hand part of the logic creation area, in the position defined when you dragged it previously.
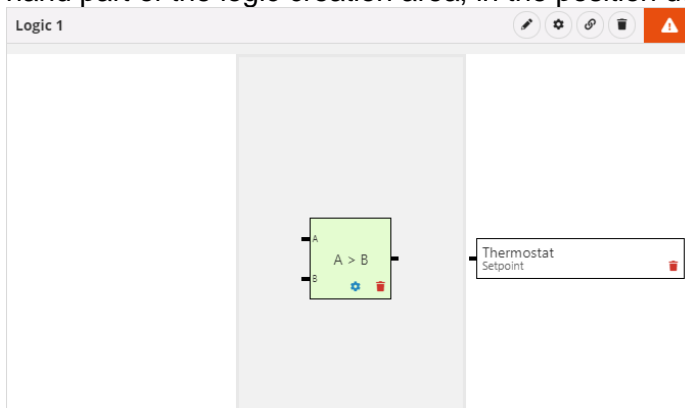
Press the ⎯ icon if you want to delete the element (even if it's already connected to an operator).

To make the connection between the logic operator and the output, select the connector that identifies the logic operator output and drag it onto the required element; during this operation, the border of the elements that are compatible with the logic operator will turn green. If an element is highlighted in green when the cursor moves onto it, this means the connection can be made.
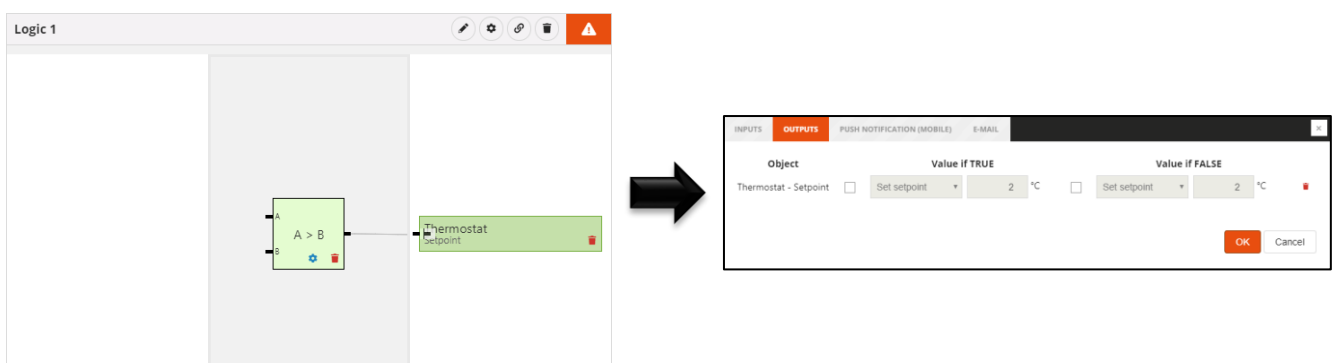
When the object is released, the logic operator settings relating to the outputs are shown:

Object → name of the element and relative command type

Value if true → if the checkbox is ticked, this allows you to set the command that will be sent when the result of the logic is TRUE; if the checkbox is disabled, there is no action on the element when the result is TRUE.

Value if false → if the checkbox is ticked, this allows you to set the command that will be sent when the result of the logic is FALSE; if the checkbox is disabled, there is no action on the element when the result is FALSE.

→ deletes the element from the logic operator output (removes the connection between the two objects)

There are no limits to the number of outputs that can be associated with a logic.

When the object is connected to the logic operator, you can remove the connection by means of the ✂ icon.
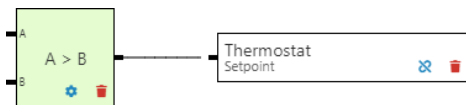
To insert a social action output, select the element (Push notification or e-mail) and drag it onto the right-hand part of the logic creation area; during this operation, a grey box will appear with the name of the element, to help you position the object in the creation area. When the element is released, it's shown (name and type of social action) in the right-hand part of the logic creation area, in the position defined when you dragged it previously.
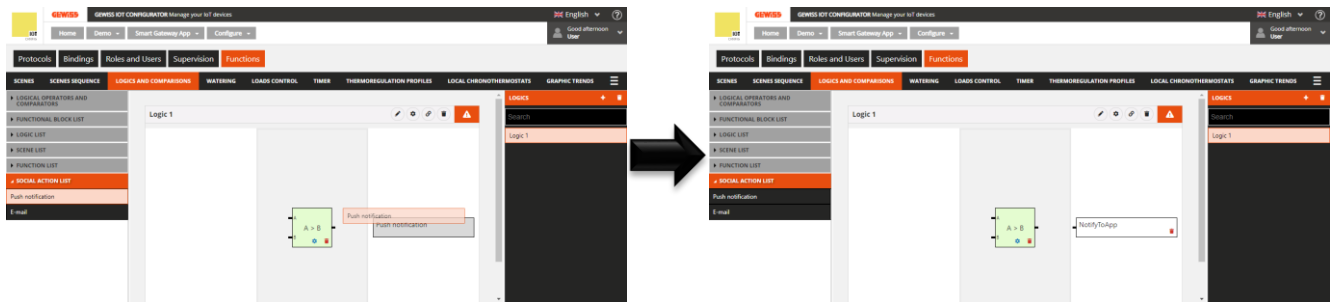
Press the 🗑 icon if you want to delete the element (even if it's already connected to an operator).

To make the connection between the logic operator and the social action, select the connector that identifies the logic operator output and drag it onto the required element; during this operation, the border of the elements that are compatible with the logic operator will turn green. If an element is highlighted in green when the cursor moves onto it, this means the connection can be made.
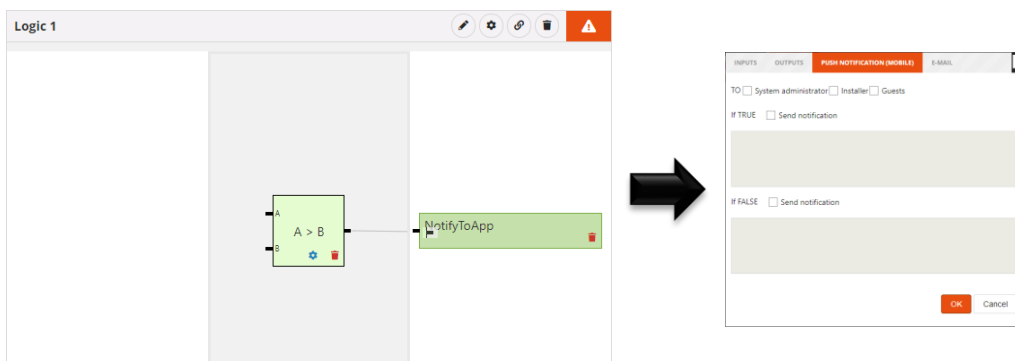

When the object is released, the logic operator settings relating to the Push notifications or e-mails (depending on which object was inserted) are shown:

PUSH NOTIFICATIONS

A → list of the project roles that the notification is addressed to

If true → if the previous checkbox is ticked, this allows you to set the text of the push notification that will be sent when the result of the logic is TRUE; if the checkbox is disabled, no notification is sent when the result is TRUE.

If false → if the previous checkbox is ticked, this allows you to set the text of the push notification that will be sent when the result of the logic is FALSE; if the checkbox is disabled, no notification is sent when the result is FALSE.

Only one "Push notification" type object can be inserted in a logic.

E-MAILS

A → list of the e-mail addresses that the message will be sent to; several e-mail addresses can be inserted, as long as they're separated with "**;**" (e.g. "**demo@gewiss.com; gewiss@gewiss.com**")

If true → if the previous checkbox is ticked, this allows you to set the text of the push notification that will be sent when the result of the logic is TRUE; if the checkbox is disabled, no notification is sent when the result is TRUE.

If false → if the previous checkbox is ticked, this allows you to set the text of the push notification that will be sent when the result of the logic is FALSE; if the checkbox is disabled, no notification is sent when the result is FALSE.

Only one "E-mail" type object can be inserted in a logic.

When the object is connected to the logic operator, you can remove the connection by means of the ✂ icon.

One or more than one operator can be linked to construct more complex logic functions. When the operators are linked, actions can only be associated with the last one in the chain. For linking purposes, all the operators are considered as Boolean inputs.

After dragging the operators into the logic configuration area, to make the connection between the two operators you must select the connector that identifies the logic operator input (the second node of the chain) and drag it onto the other operator (the first of the chain); during this operation, the border of the elements that are compatible with the logic operator will turn green.
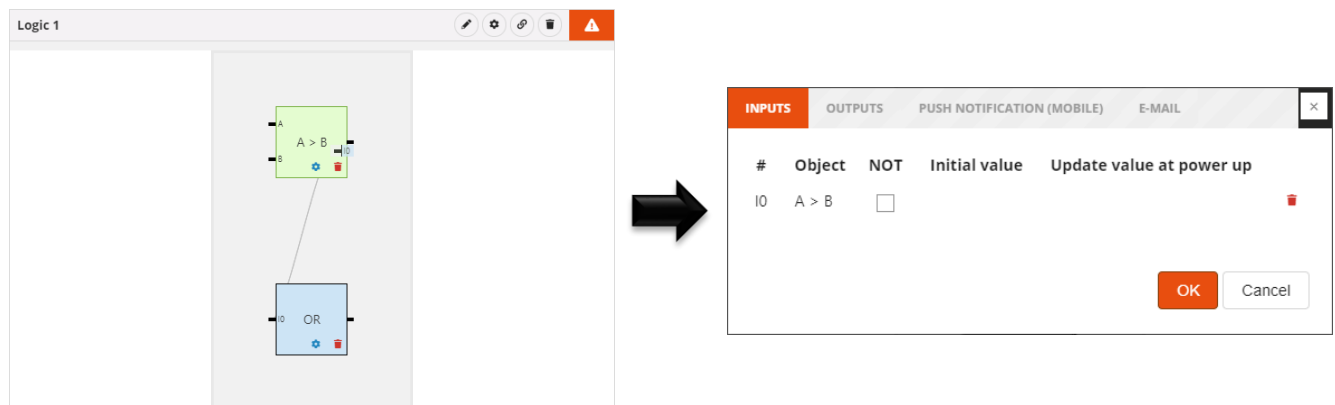
When the object is released, the logic operator settings (second node of the chain) relating to the inputs are shown:

→ name of the connector that the element is connected to

Object → name of the element

Not → enables the refusal of the result of the logic operator input; when the value of an input is refused, the connector is preceded by a white spot (see "I0" in the example below)


Initial value → no setting available

Update data item at switch-on → no setting available

→ deletes the element from the logic operator input (removes the connection between the two objects)

ATTENTION: an incomplete logic isn't saved on the cloud. This means that it will no longer be present when the page is changed, or when the project is closed and reopened, even if the configuration has been sent to the area shared with clients (by pressing the 🔵 push-button).

All the valid logic functions created are listed in alphabetical order in the LOGIC FUNCTIONS column (on the right).

To remove a logic function from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the logic functions simultaneously by pressing the 🗑 icon in the right-hand column ("LOGIC FUNCTIONS").
ATTENTION: a logic function can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the logic function won't be deleted.
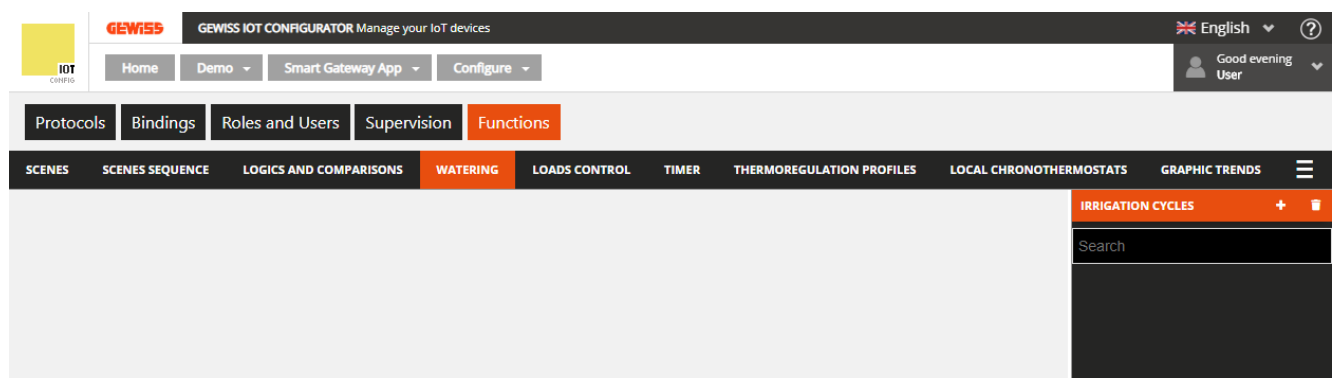
Irrigation

This function is used to define an activation sequence (cycle) for the sprinklers that are usually grouped together in separate zones of the garden.
Each irrigation cycle can be activated independently of the others, and associated with the days of the week and an activation time.
Once you have selected the "Functions" menu on the first row and the "IRRIGATION" menu on the second, the IRRIGATION CYCLES column will appear (on the right).

To create a new irrigation cycle, press the "**+**" icon in the right-hand column ("IRRIGATION CYCLES").
Enter the name you want to associate with the irrigation cycle, then confirm.
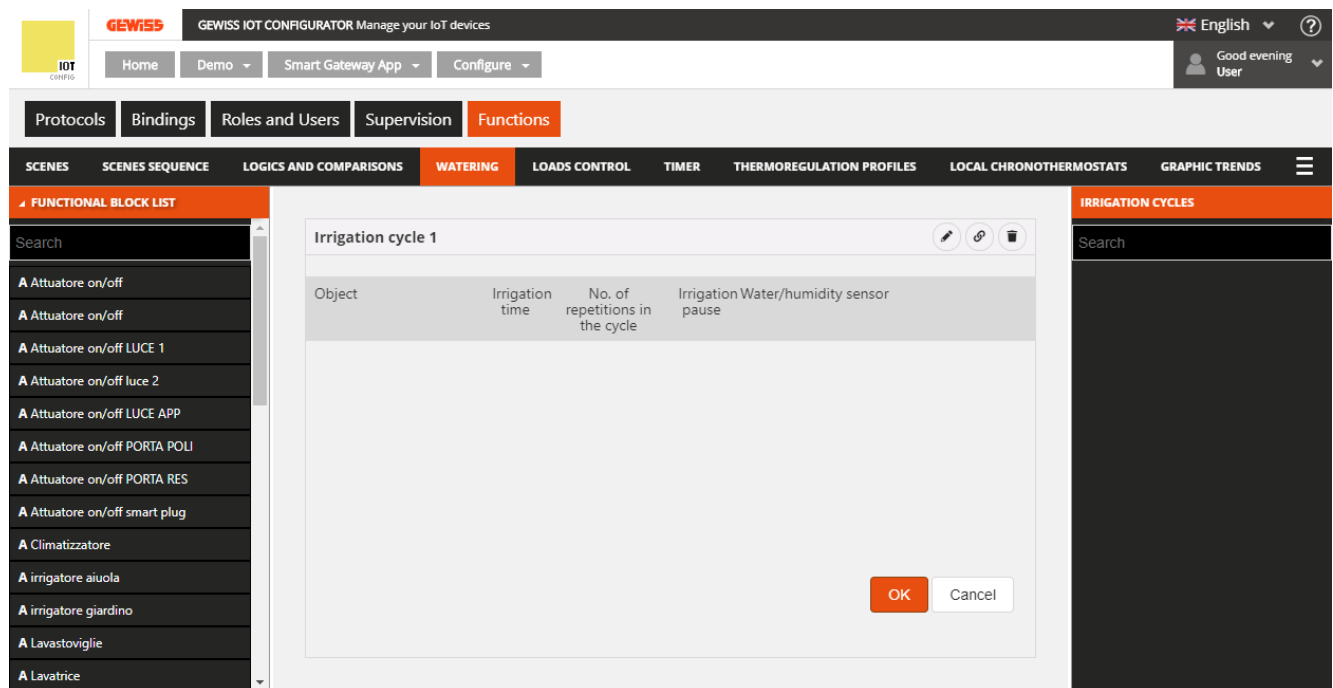
The central part of the page will show a list of all the sprinklers - and the relative parameters - involved in the irrigation cycle.

Next to the name of the irrigation cycle, you can see:

🖉 **Rename** push-button for modifying the name of the cycle

🔗 **View connections** push-button for displaying all the pages where the cycle is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the cycle

The left-hand column (LIST OF FUNCTIONAL BLOCKS) shows all the Actuator type functional blocks, both ZigBee and KNX, that can receive on/off commands (e.g. Actuator on/off).

All the functional blocks are shown with their name and category ("A" = Actuator).
At the head of the list there's a text search field for filtering the contents.
The functional blocks already used in the irrigation cycle you are editing are highlighted in grey.

Select a functional block and drag it into the central area in order to add it to the irrigation cycle you are editing; release the object in the list of sprinklers, in the required activation position. At the end of this operation, a pop-up menu will appear where you can select:
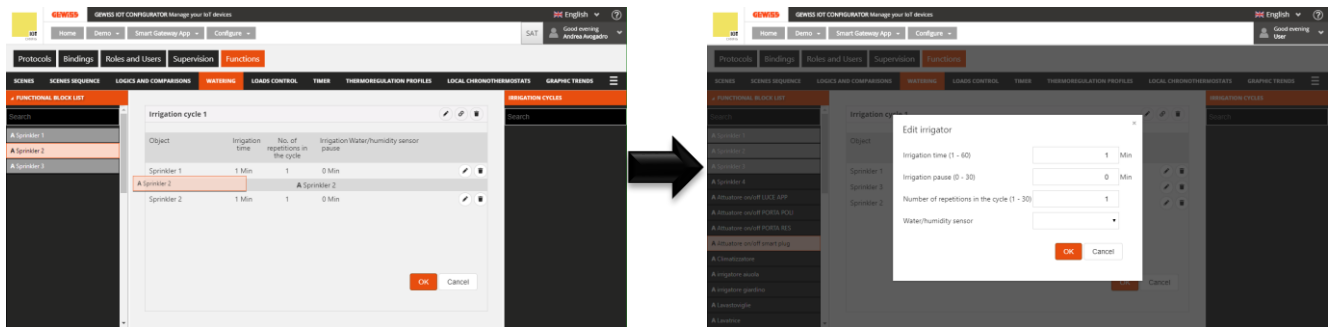
- Sprinkler activation time. Possible values: from 1 to 60
- Pause before the activation of the next sprinkler. Possible values: from 1 to 30
- Number of sprinkler activations during the irrigation cycle. Possible values: from 1 to 30
- Selection of the optional humidity/rain sensor that inhibits the sprinkler.

After confirming the settings, the command will be visualised in the central part of the page, in the list of sprinklers in the cycle, in the position defined when you dragged it previously.
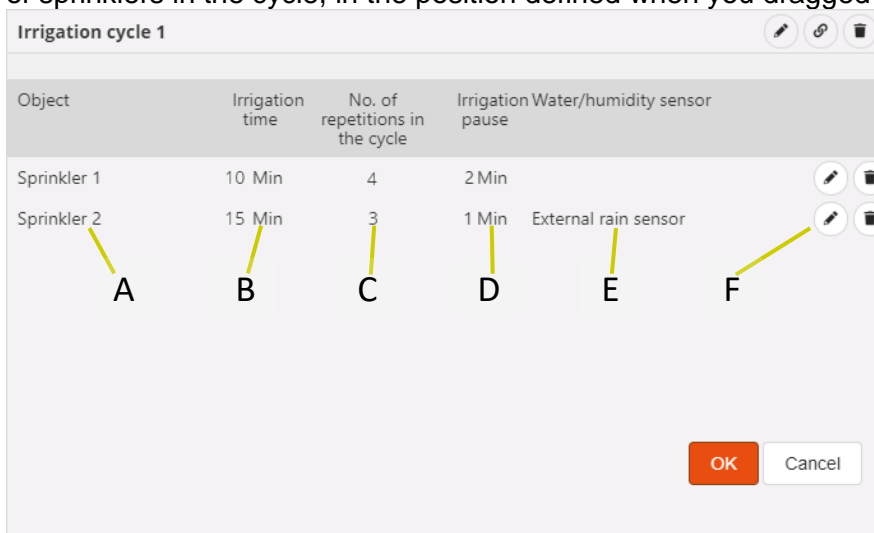
A. Sprinkler name
B. Sprinkler activation time
C. Number of sprinkler activations during the irrigation cycle
D. Pause before the activation of the next sprinkler
E. Rain/humidity sensor that inhibits the sprinkler
F. Contextual functions

  ✏ Modify the sprinkler parameters (the same ones visualised when the object was being inserted in the cycle); this function is called up by double clicking on the row that represents the sprinkler

  🗑 Delete the sprinkler from the list

To move a sprinkler within the list, just select it and then drag it to the required position.

There are no limits to the number of sprinklers that can be associated with a cycle.

Press the "OK" push-button to see the weekly programming for irrigation cycle activation. NOTE: this push-button is only enabled when there is at least one sprinkler in the list.

Below the cycle name, there is the parameter "Irrigation time % variation", which allows to set the percentage of increase of water supply with respect to the scheduled times. Modifying this value, the cycle end times will be recalculated too, with consequent update of the profile. Example: by setting "-20%", the duration of the irrigation cycle and all the set times will be reduced by 20%. This value may eventually also be changed by the App.

Underneath the parameter, there's a time line (00-24). To alter the time resolution, bring the cursor inside the programming grid and rotate the mouse wheel upwards (to increase the resolution) or downwards (to reduce it). When the resolution doesn't allow you to see the entire 00-24 scale, you can move along the time axis by selecting any point on it and dragging it to the right or left.

Each row represents one day of the week.

To the left of the name of the day, you can see the enabling status for that day's programming (orange - disabled, green - enabled). Click on the status to change the value.

To repeat the programming of one day on another day, press the ⧉ icon to the right of the day whose programming you want to repeat; from the pop-up menu that appears, select the day on which you want to copy the programming, then confirm.

To enable/disable the programming of a day of the week, press the coloured band to the left of the name:

 Programming enabled

 Programming disabled

On the time line for each day, there's a block that represents the total duration of the irrigation cycle.

Inside the block, you can see:
- the cycle activation time, preceded by the ▶ icon
- the irrigation cycle end time (calculated as the total of the various irrigation times), preceded by the ■ icon
- ✎ push-button for modifying the cycle activation time

The cycle activation time can be altered directly from the time grid: select the required block (it will turn yellow), then select the left-hand edge of the block and drag it to the right or left to change the activation time in 15-minute steps (the whole block will move). Release to save the modification.

NOTE: the implementation of an irrigation cycle associated with a specific time period (calendar) isn't defined in programming; it's one of the cycle activation options (disabled, enabled, enabled with calendar) managed via the app. Forced activation, and the % increase of water supply on the basis of the value of the programmed cycle, are also managed via the app (not in programming).

When the weekly programming of irrigation cycle activation is shown in the central area, you will see a new icon ⚙ next to the name of the irrigation cycle: press this icon to see the list of sprinklers (described above).
Every time you select an irrigation cycle from the IRRIGATION CYCLES column, the weekly programming of cycle activation will be visualised; to modify the list of sprinklers or the various irrigation times, press the ⚙ icon next to the name of the cycle.

All the irrigation cycles created are listed in alphabetical order in the IRRIGATION CYCLES column (on the right).
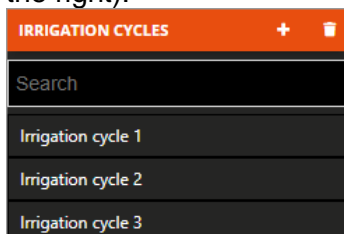
To remove an irrigation cycle from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the irrigation cycles simultaneously by pressing the 🗑 icon in the right-hand column ("IRRIGATION CYCLES").
ATTENTION: an irrigation cycle can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the cycle won't be deleted.

Load control

The load control function is used to manage the activation/deactivation of electric loads connected to the actuators, to prevent the electricity counter from disconnection if the contractual power level is exceeded.

By monitoring the instantaneous power and the power threshold values, the system generates the gradual disconnection of the loads until the set threshold value is reached.

The control of each load group can be activated independently of the others, and associated with the days of the week and an activation time.

Once you have selected the "Functions" menu on the first row and the "LOAD CONTROL" menu on the second, the LOAD CONTROL column will appear (on the right).
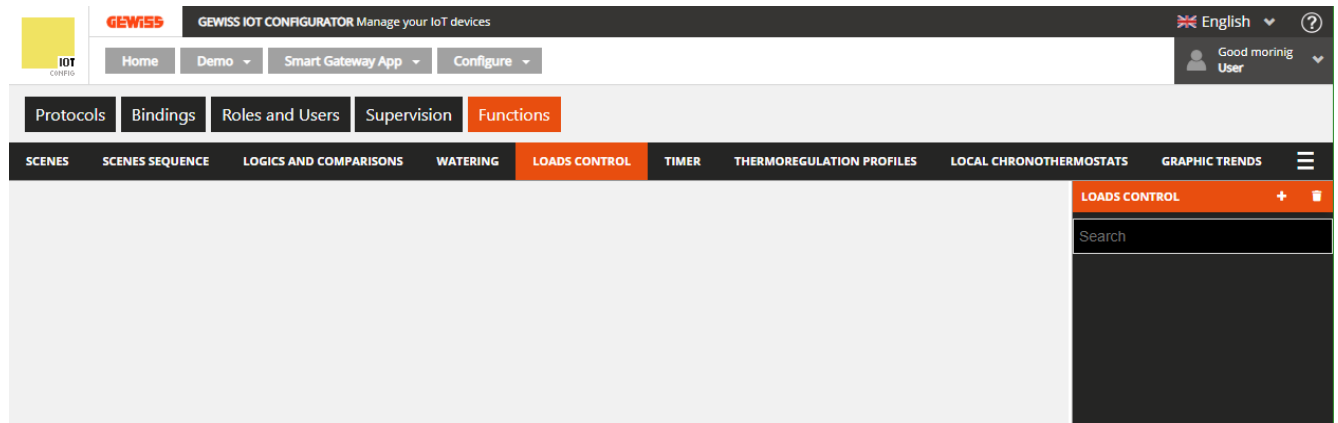
To create a new group of loads, press the "**+**" icon in the right-hand column ("LOAD CONTROL").
Enter the name you want to associate with the group of loads, then confirm.
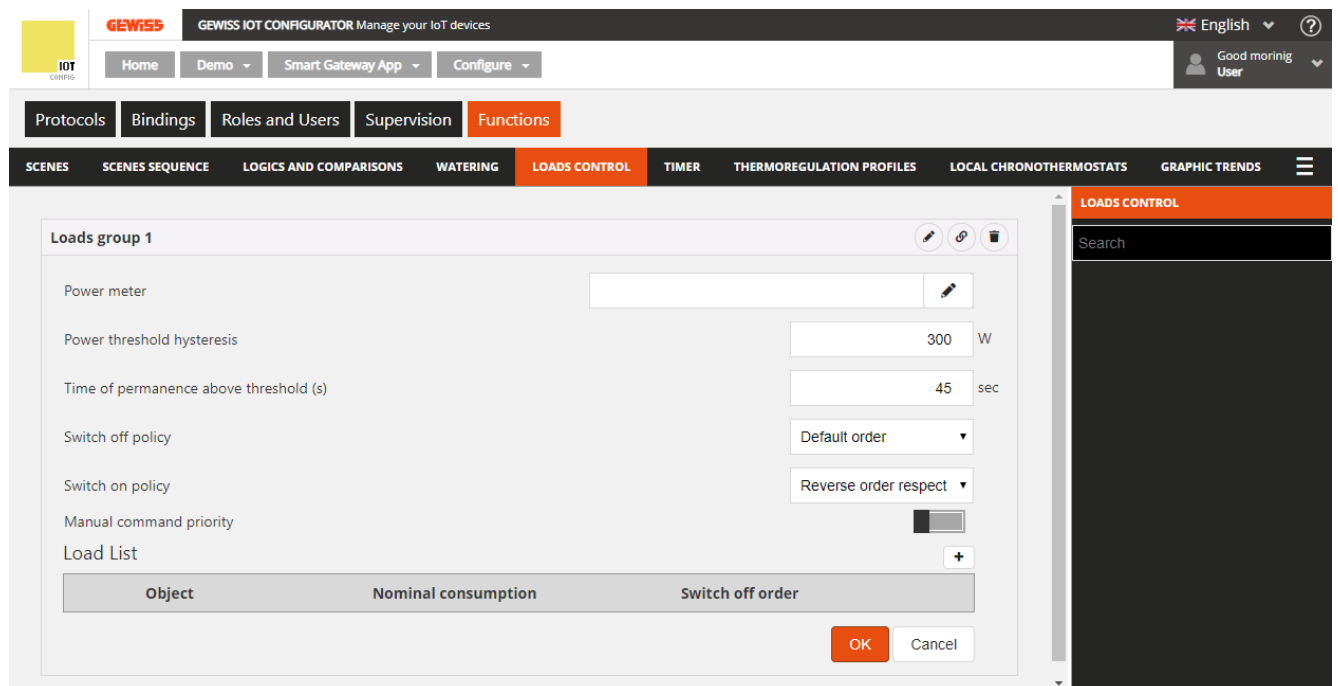
Next to the name of the group of loads, you can see:

Rename push-button for modifying the name of the group

View connections push-button for displaying all the pages where the group is used (select the required page to gain direct access)

Delete push-button for eliminating the group

When this function is active, the device begins monitoring the instantaneous active power sent by the gauge and, if the level exceeds the "power threshold + hysteresis" value, it starts the count of the time of persistence before intervening by disconnecting the loads. At the end of the count time (if the power hasn't fallen below the "power threshold + hysteresis" value), the disconnection of the loads begins on the basis of the rule set; the loads are disconnected at 5-second intervals from each other. The disconnection ends when the power falls below the threshold value. Once this condition has been reached, the device begins evaluating the reconnection of the loads.

The reconnection takes place on the basis of the rule set; the loads are reconnected at 5-second intervals from each other. During the reconnection process, the instantaneous power value and nominal absorption (or real load absorption, if available) are evaluated (at the moment of disconnection): if the total of the two values means the disconnection threshold may be exceeded, then load isn't reconnected but the reconnection of the next load is considered (in reconnection order).
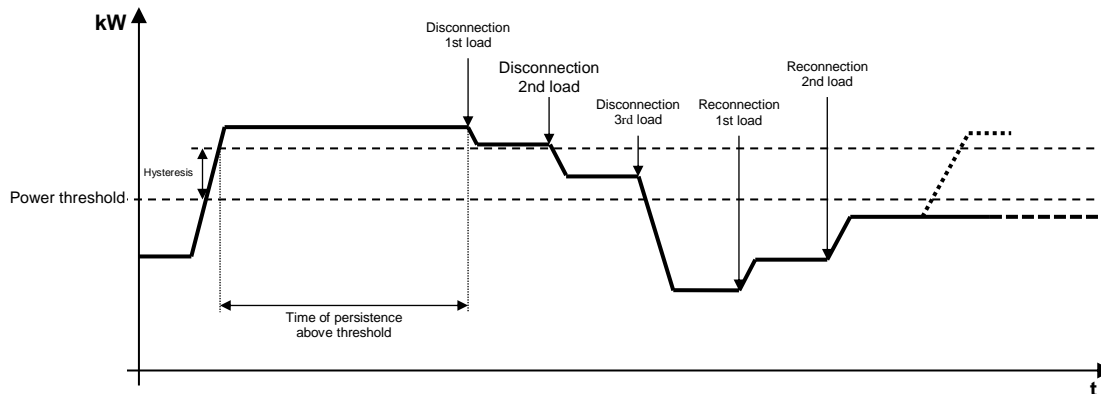
The central part of the page will show the parameters that determine the behaviour of the control algorithm, and the list of all the loads - and relative parameters - involved in the control.

The parameters to be set are:

- Selection of the power gauge that will supply the value to be monitored. Press the ✏ icon to see the pop-up with a list of the functional blocks compatible with the function.
- Hysteresis of the power threshold value. Possible values: from 10 to 500W
- Time of persistence of the power value above the threshold, before beginning the disconnection. Possible values: from 10 to 240 seconds
- Load disconnection rule. Possible values:
  1. Predefined order: the disconnection is made in the order defined in the list of loads
  2. From the load that absorbs the most: the disconnection begins with the load that's absorbing the most, down to the one that's absorbing the least
  3. From the load that absorbs the least: the disconnection begins with the load that's absorbing the least, down to the one that's absorbing the most
- Load reconnection rule. Possible values:
  1. Same order as for disconnection: reconnection follows the same order used for disconnection (FIFO logic - the first load disconnected will be the first to be reconnected)
  2. Reverse order compared with disconnection: reconnection follows the reverse of the order used for disconnection (LIFO logic - the last load disconnected will be the first to be reconnected)
  3. No automatic reconnection: the reconnection isn't performed automatically by the algorithm, but must be implemented directly by the user (via the app or from another command point)
- Enabling of the priority of the manual command over the command sent by the disconnection algorithm. If this option is enabled, the Smart Gateway ignores the fact that the load (which, for the load control algorithm, must be disconnected) has been reconnected from another command point, thereby avoiding reconnecting it. Vice versa, if the option is disabled, the Smart Gateway disconnects the load again if it's reconnected from another command point.

To add a load to the group you are editing, press the"+" icon above the list of loads. A pop-up menu will then appear, where you can select:

- the functional block to be added to the list (all the Actuator type functional blocks, both ZigBee and KNX, that can receive on/off commands (e.g. Actuator on/off) and aren't already in the list)
- nominal consumption of the load (used if the instantaneous absorption of the physical device isn't available). Possible values: from 10 to 3000W

Once the settings have been confirmed, the load will be shown in the central part of the page, within the list of group loads.

Object	Nominal consumption	Switch off order		
Washing machine	1800 W	1	✏	🗑
Dishwasher	1200 W	2	✏	🗑

Load List +

A   B   C   D

A. name of the load
B. nominal consumption
C. disconnection order (only visible if the disconnection rule set is "Predefined order")
D. contextual functions

✏ Modify the load parameters (the same ones visualised when the load was being inserted in the group)

🗑 Delete the load from the list

To move a load within the list, just select it and then drag it to the required position.

There are no limits to the number of loads that can be associated with a group.

Press the "OK" push-button to see the weekly programming of the power threshold associated with the group.

At the top there's a time line (00-24). To alter the time resolution, bring the cursor inside the programming grid and rotate the mouse wheel upwards (to increase the resolution) or downwards (to

reduce it). When the resolution doesn't allow you to see the entire 00-24 scale, you can move along the time axis by selecting any point on it and dragging it to the right or left.

Each row represents one day of the week.

To add a new time band in a specific day, press the **+** icon on the right of the name of the day concerned. In the pop-up menu that appears, select the band start time and the threshold value the band end time coincides with the start time of the next band, or with the end of the day.

To repeat the programming of one day on another day, press the 📑 icon to the right of the day whose programming you want to repeat; from the pop-up menu that appears, select the day on which you want to copy the programming, then confirm.

On the time line for each day, there are the time bands with the start time and relative power threshold value. Daily programming can never be blank - there must be at least one band (the band that begins at 00:00 can't be deleted).

3000W
08:00 ✏ 🗑
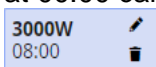
Inside the band, you can see:
- the value of the power threshold in that time band
- the band start time (underneath the power value)
- ✏ push-button for modifying the band start time and threshold value
- 🗑 push-button for deleting the band

The start and end times of a band can be modified directly from the time grid: select the required band (it will turn yellow), then select the left-hand edge of the band and drag it to the right or left to change the start time in 15-minute steps, or select and drag the right-hand edge to change the end time. Release to save the modification.

NOTE: the implementation of the load control algorithm associated with a specific time period (calendar) isn't defined in programming; it's one of the algorithm activation options (disabled, enabled, enabled with calendar) managed via the app.

When the central area shows the weekly programming of the power threshold, a new icon ⚙ will appear next to the name of the load group, along with the validity status of the element.

⚠ ✅ Icon indicating the validity status of the load group: orange if it's incomplete and can't therefore be used in the project, green if it's complete and can be used. If the group is incomplete, click on the orange icon to see the type of error.

⚙ Press this icon to see the parameters that determine the behaviour of the control algorithm and the list of loads (described above).

ATTENTION: an incomplete group isn't saved on the cloud. This means that it will no longer be present when the page is changed, or when the project is closed and reopened, even if the configuration has been sent to the area shared with clients (by pressing the 🔵 push-button).

Every time you select a load group from the LOAD CONTROL column, the weekly programming of the group power threshold will be visualised; to modify the list of loads or the algorithm parameters, press the ⚙ icon next to the name of the group.

All the load groups created are listed in alphabetical order in the LOAD CONTROL column (on the right).

To remove a load group from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the groups simultaneously by pressing the 🗑 icon in the right-hand column ("LOAD CONTROL").

ATTENTION: a load group can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the group won't be deleted.

Daily/Weekly timer

A timer identifies a daily/weekly profile associated with a specific action (light command, scene application, roller shutter command, etc.) to automate periodic actions; For instance, the slates can be opened/closed in defined periods to protect the room from sunlight, or the "wake-up" scene can be activated every day at the same time.

The Smart Gateway manages the programmed implementation of the actions according to the settings configured.

Once you have selected the "Functions" menu on the first row and the "TIMER" menu on the second, the TIMER column will appear (on the right).
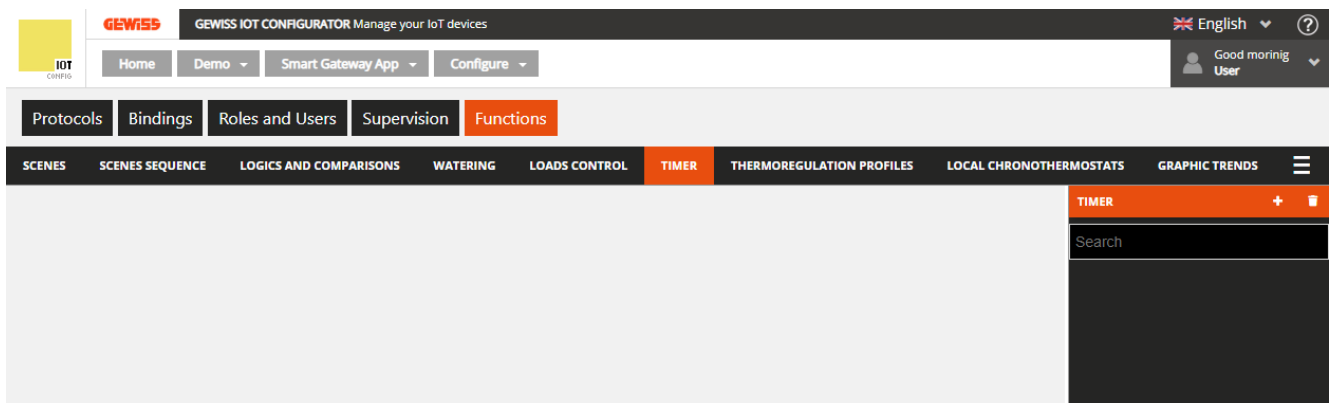
To create a new timer, press the "+" icon in the right-hand column ("TIMERS").

In the pop-up menu that appears, enter the name and

- the type of time profile (Timer type)
  a. Daily → the time profile is a daily one, so the actions can be programmed for a single day

b. Weekly → the time profile is a weekly one, so the actions can be programmed for all 7 days of the week
- selection of the functional block/scene/scene sequence/logic rule that you want to associate the time profile with
- selection of the type of action of the selected element that you want to control with the profile (Object); the options will depend on the element you've selected
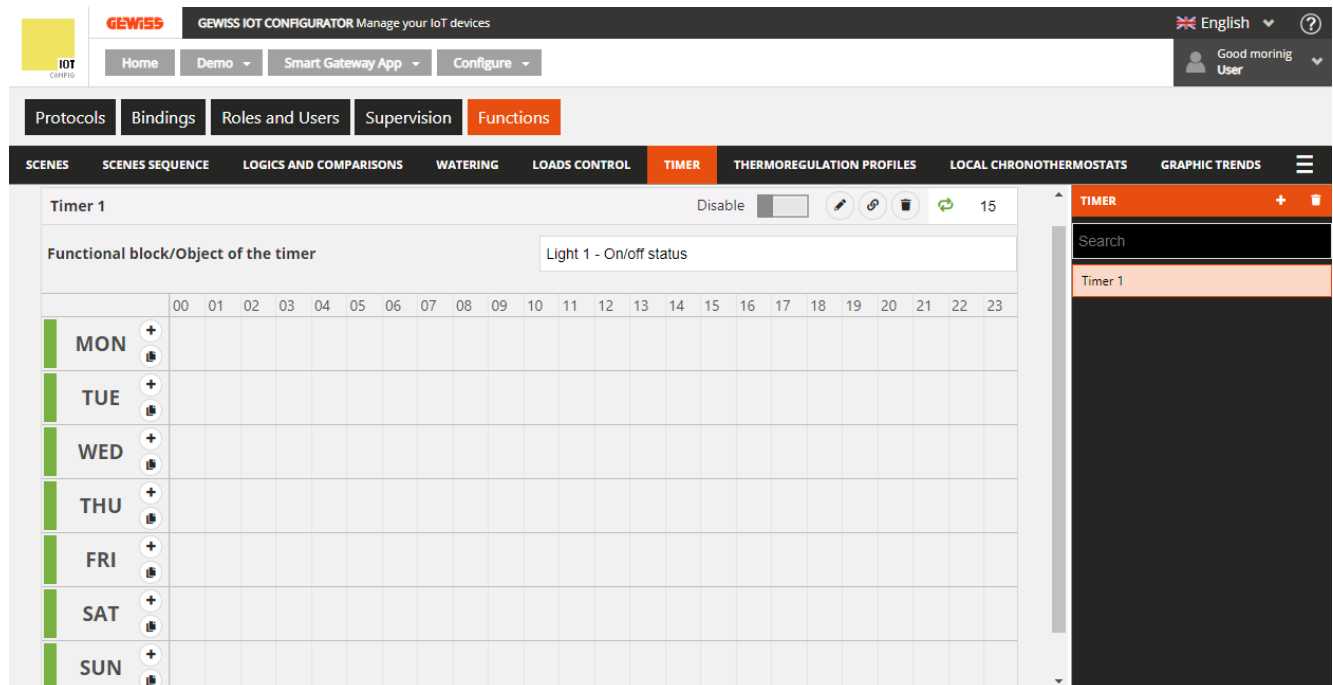
Next to the name of the timer, you can see:

Rename push-button for modifying the name of the timer

View connections push-button for displaying all the pages where the timer is used (select the required page to gain direct access)

Delete push-button for eliminating the timer

Numerical field for setting the command repeat period for the time bands on which value repetition is enabled. Possible values: from 1 to 45 minutes

The central part of the page will show the weekly (or daily) programming of the object selected.
At the top there's a time line (00-24). To alter the time resolution, bring the cursor inside the programming grid and rotate the mouse wheel upwards (to increase the resolution) or downwards (to reduce it). When the resolution doesn't allow you to see the entire 00-24 scale, you can move along the time axis by selecting any point on it and dragging it to the right or left.

If the profile is weekly, each row represents one day of the week; if it's daily, there is only one row.

To add a time band in a specific day, press the **+** icon on the right of the name of the day concerned.
From the pop-up menu that appears, select:
- the band start time
- the enabling of command repetition within the time band (if enabled, the set value will be repeated cyclically until the end of the band)
- the command to send
the band end time coincides with the start time of the next band, or with the end of the day.

To repeat the programming of one day on another day, press the 📑 icon to the right of the day whose programming you want to repeat; from the pop-up menu that appears, select the day on which you want to copy the programming, then confirm.

To enable/disable the programming of a day of the week, press the coloured band to the left of the name:

MON ⊕ 📑   Programming enabled

MON ⊕ 📑   Programming disabled

On the time line for each day, there are the time bands with the start time and relative value to be sent to the functional block.

14:20 ✏ 🗑
On
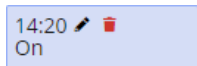
Inside the band, you can see:
- the band start time
- the command to send to the functional block selected (underneath the band start time)
- ✏ push-button for modifying the start time, the value to send, and the command repetition
- 🗑 push-button for deleting the band

If value repetition has been enabled within the time band, the background will be light blue; if the value isn't repeated, the background remains white.

The time band start time can be altered directly from the time grid: select the required band (it will turn yellow), then select the left-hand edge of the band and drag it to the right or left to change the start time in 5-minute steps. Release to save the modification.

NOTE: the implementation of a timer associated with a specific time period (calendar) isn't defined in programming; it's one of the timer activation options (disabled, enabled, enabled with calendar) managed via the app.

All the timers created are listed in alphabetical order in the TIMERS column (on the right).

TIMER	＋	🗑
Search		
Timer 1		
Timer 2		
Timer 3		
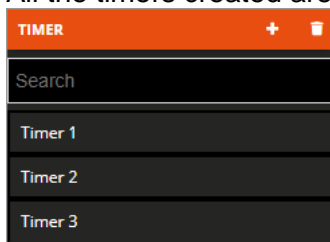
To remove a timer from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the timers simultaneously by pressing the 🗑 icon in the right-hand column ("TIMERS").

ATTENTION: a timer can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the timer won't be deleted.

Temperature adjustment profiles

One particular timer function is the temperature adjustment profile. This profile sends (weekly and hourly - 0-24) the HVAC mode or temperature setpoint
to the thermostats or temperature adjustment probes configured as Slaves. Specific profiles for heating/cooling can be defined.
The Smart Gateway manages the programmed implementation of the actions according to the settings configured.
Once you have selected the "Functions" menu on the first row and the "TEMPERATURE ADJUSTMENT PROFILES" menu on the second, the THERMOSTATS column will appear (on the right).

To create a new temperature adjustment profile, press the "**+**" icon in the right-hand column ("THERMOSTATS").

Select the functional block whose hourly profile you want to activate (the list shows all the "HVAC thermostat" or "Setpoint thermostat" functional blocks for which no temperature adjustment profile is active.
Press "OK" to see the weekly programming of the temperature adjustment profile associated with the functional block selected.

Next to the name of the functional block whose profile has been activated, you can see:

✏️ **Rename** push-button for modifying the name of the temperature adjustment profile

🔗 **View connections** push-button for displaying all the pages where the profile is used (select the required page to gain direct access)

🗑️ **Delete** push-button for eliminating the profile

🔁 1 Numerical field for setting the command repeat period within the time bands. Possible values: from 1 to 45 minutes

Underneath the name of the profile, there are two push-buttons for selecting the profile to be programmed: heating or cooling; in fact, two separate profiles are reserved for each functional block for the two types of operation. The Smart Gateway sends the correct commands based on the type of operation active on the controlled device.

The central part of the page will show the weekly programming of the functional block selected.
At the top there's a time line (00-24). To alter the time resolution, bring the cursor inside the programming grid and rotate the mouse wheel upwards (to increase the resolution) or downwards (to reduce it). When the resolution doesn't allow you to see the entire 00-24 scale, you can move along the time axis by selecting any point on it and dragging it to the right or left.
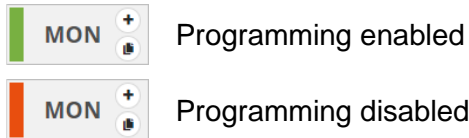
Each row represents one day of the week.

To add a new time band in a specific day, press the **+** icon on the right of the name of the day concerned. In the pop-up menu that appears, select the band start time and the threshold value (HVAC or setpoint mode, depending on the type of functional block); the band end time coincides with the start time of the next band, or with the end of the day.

To repeat the programming of one day on another day, press the 📄 icon to the right of the day whose programming you want to repeat; from the pop-up menu that appears, select the day on which you want to copy the programming, then confirm.

To enable/disable the programming of a day of the week, press the coloured band to the left of the name:

MON   Programming enabled

MON   Programming disabled

On the time line for each day, there are the time bands with the start time and relative value to be sent. Daily programming can never be blank - there must be at least one band (the band that begins at 00:00 can't be deleted).

20°C
17:00

Inside the band, you can see:
- the value (HVAC or setpoint mode) to be sent in that time band
- the band start time (underneath the value to be sent)
- ✏ push-button for modifying the start time and the value to send
- 🗑 push-button for deleting the band

If the profile is "heating", the various time bands are coloured in different shades of red according to their value; the same for the "cooling" profile, but in blue.

The start and end times of a band can be modified directly from the time grid: select the required band (it will turn yellow), then select the left-hand edge of the band and drag it to the right or left to change the start time in 15-minute steps, or select and drag the right-hand edge to change the end time. Release to save the modification.

NOTE: the implementation of a temperature adjustment profile associated with a specific time period (calendar) isn't defined in programming; it's one of the profile activation options (disabled, enabled, enabled with calendar) managed via the app. Once activated, the associated profile will be activated/visualised depending on whether the Slave device is in heating or cooling mode (without the user having to select it).

All the temperature adjustment profiles created are listed in alphabetical order in the THERMOSTATS column (on the right).

THERMOSTATS   +   🗑
Search
Thermostat - living room
Thermostat - bed room
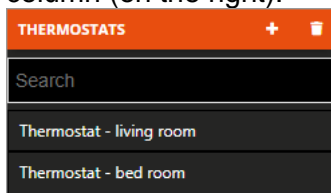
To remove a profile from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the temperature adjustment profiles simultaneously by pressing the 🗑 icon in the right-hand column ("THERMOSTATS").
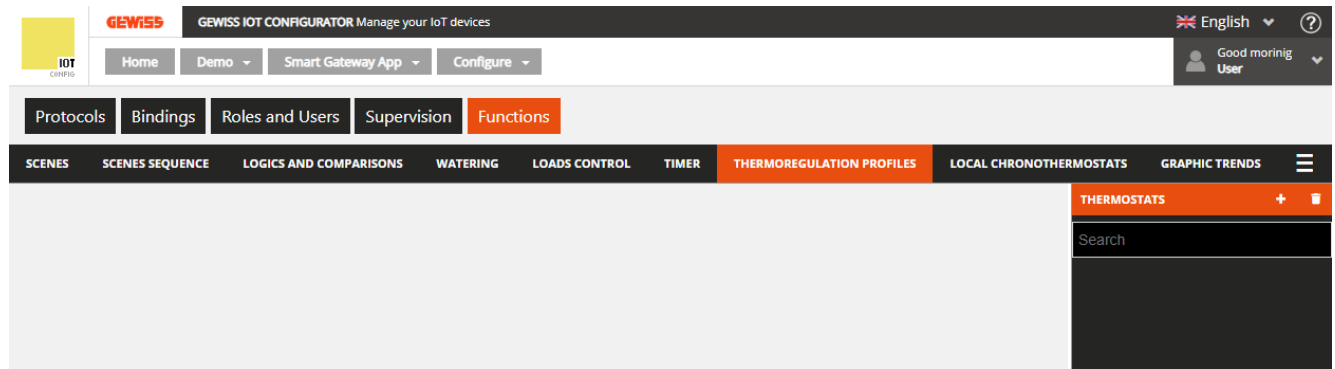
ATTENTION: a temperature adjustment profile can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the profile won't be deleted.

Local timed thermostats

The Smart Gateway implements the temperature adjustment control logic directly, with the possibility to receive the temperature from an external sensor, parametrize setpoints and regulation differentials, and then regulate the valves and/or fancoils as necessary.

The Gateway doesn't have its own temperature sensor. The integrated control function relates to the possibility to manage temperature adjustment (via a linked temperature adjustment profile), but using temperature values received from an external sensor.

Once you have selected the "Functions" menu on the first row and the "LOCAL TIMED THERMOSTATS" menu on the second, the TIMED THERMOSTATS column will appear (on the right).
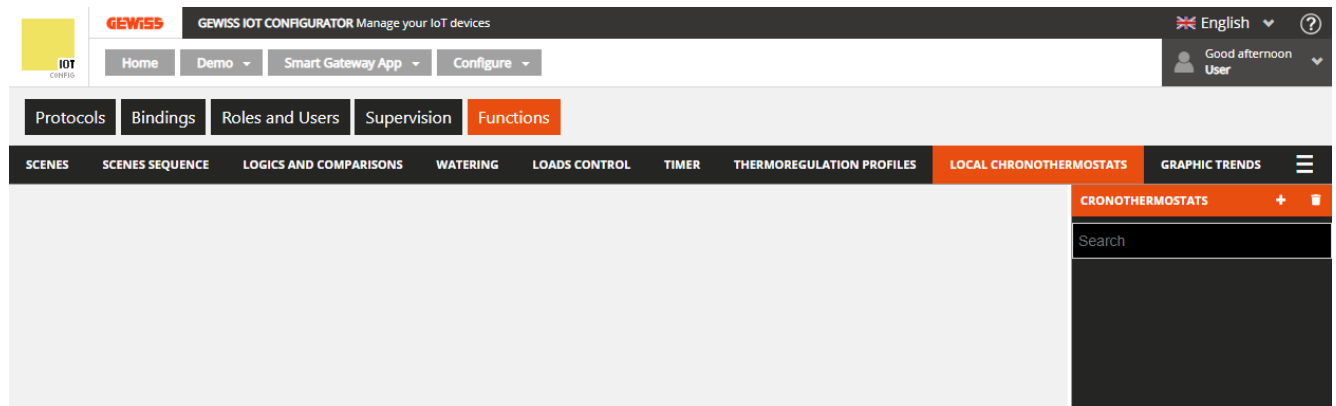
To create a new timed thermostat, press the "**+**" icon in the right-hand column ("TIMED THERMOSTATS").

Enter the name you want to associate with the timed thermostat, then confirm.
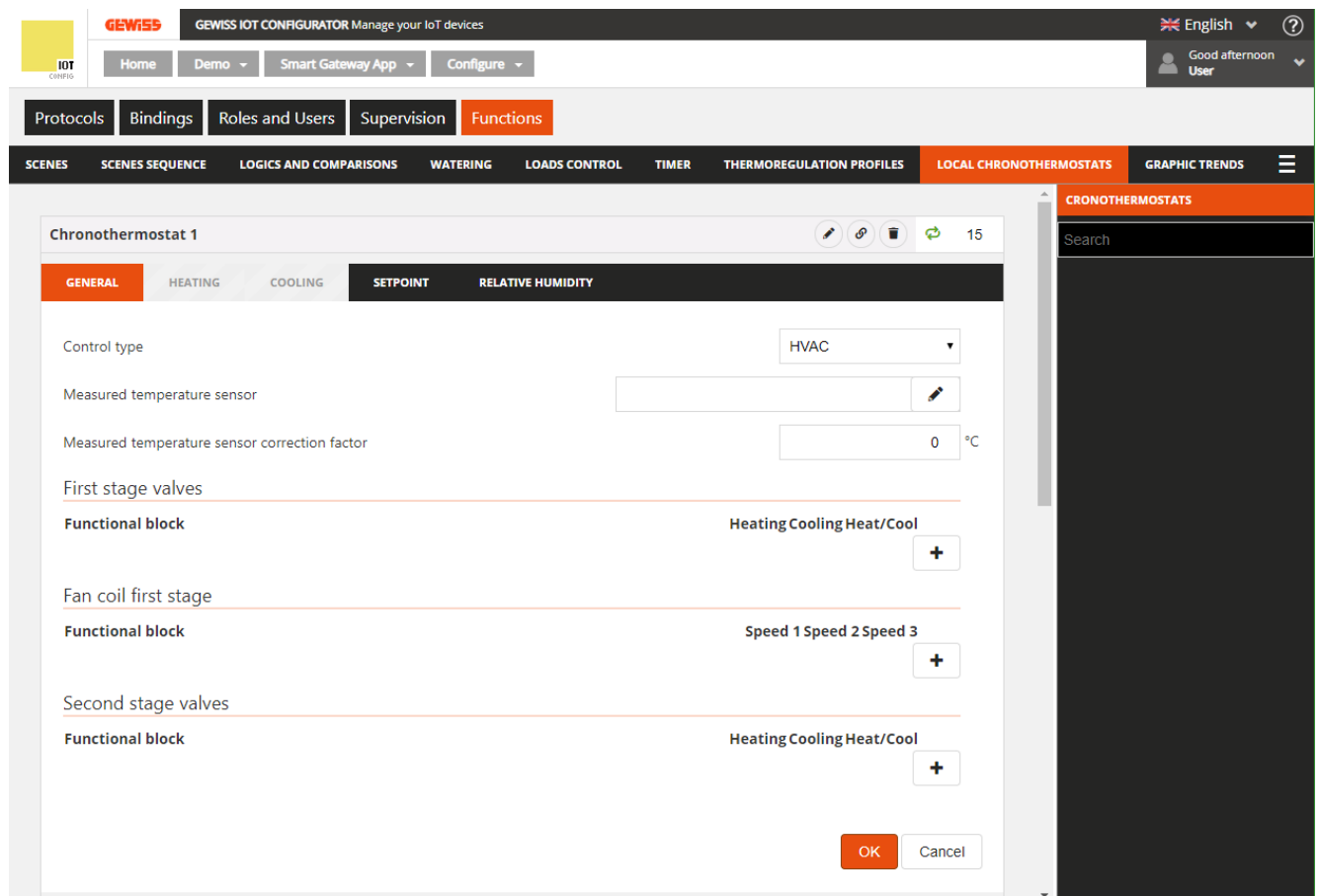
Next to the name of the timed thermostat, you can see:

✏ **Rename** push-button for modifying the name of the timed thermostat

🔗 **View connections** push-button for displaying all the pages where the timed thermostat is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the timed thermostat

The central part of the page will show the parameters that determine the behaviour of the timed thermostat, sub-divided into various sub-sections.
In the GENERAL section, the parameters to be set are:
- The type of timed thermostat control (HVAC or setpoint), which determines the general operation of the thermostat (the choice also affects the parameters that will be visible in the SETPOINT section)
- Selection of the temperature sensor that will provide the value to be monitored; press the ✏ icon to see the pop-up with a list of the functional blocks compatible with the function (and not already used in the other timed thermostats created)
- Static correction of the temperature value received from the temperature sensor (if it's influenced by other factors). Possible values: from -5°C to 5°C
- List of the implementation functional blocks of the valves of the first stage control algorithm (heating and/or cooling). To add a valve implementation functional block, press the "+" icon in the "First stage valves" area; a pop-up will appear with a list of all the Actuator type functional blocks (both ZigBee and KNX) that can receive on/off commands (e.g. Actuator on/off) or General actuator commands (with on/off commands or a percentage value) and aren't already in the list of valves or the list of fancoil speeds for the timed thermostat you are editing, or for others already created. Once the choice has been confirmed, the functional block is shown in the list with three adjacent check boxes for selecting the functional block function: heating and cooling valve control (2-way system), heating valve control (4-way system or heating-only system) or cooling valve control (4-way system or cooling-only system).

First stage valves

Functional block	Heating	Cooling	Heat/Cool	
Heating/Cooling valve	○	○	◉	✖

＋

The list must contain at least one functional block, but maximum two.
When two functional blocks are present, the "Heat/Cool" option can't be selected.
To remove a functional block, press the ✖ icon on the right-hand side of that row.
The control algorithms that can be selected for heating and cooling will depend on the type of valve inserted (refer to the dedicated sections).
- List of the implementation functional blocks of the fancoil speeds of the first stage control algorithm (heating and/or cooling). To add a fancoil speed implementation functional block, press the "+" icon in the "First stage fancoils" area; a pop-up will appear with a list of all the Actuator type functional blocks (both ZigBee and KNX) that can receive on/off commands (e.g. Actuator on/off) or General actuator commands (with on/off commands or a percentage value) and aren't already in the list of valves or the list of fancoil speeds for the timed thermostat you are editing, or for others already created. Once the choice has been confirmed, the functional block is shown in the list with three adjacent check boxes for indicating the speed controlled by the functional block.

Fan coil first stage

Functional block	Speed 1	Speed 2	Speed 3	
Fancoil speed 1	◉	○	○	✖

＋

In order for you to be able to select the "fancoil" algorithm for heating and/or cooling, there must be three functional blocks in the list; otherwise, the algorithm can't be selected.
To remove a functional block, press the ✖ icon on the right-hand side of that row.

- List of the implementation functional blocks of the valves of the second stage control algorithm (heating and/or cooling). To add a valve implementation functional block, press the "+" icon in the "Second stage valves" area; a pop-up will appear with a list of all the Actuator type functional blocks (both ZigBee and KNX) that can receive on/off commands (e.g. Actuator on/off) and aren't already in the list of valves or the list of fancoil speeds for the timed thermostat you are editing, or for others already created. Once the choice has been confirmed, the functional block is shown in the list with three adjacent check boxes for selecting the functional block function: heating and cooling valve control (2-way system), heating valve control (4-way system or heating-only system) or cooling valve control (4-way system or cooling-only system).

Second stage valves

Functional block	Heating	Cooling	Heat/Cool	
Heating valve 2nd stage	◉	○	○	✖
				+

The list can contain no more than two functional blocks.
When two functional blocks are present, the "Heat/Cool" option can't be selected.
To remove a functional block, press the ✖ icon on the right-hand side of that row.

In the HEATING section (that can be accessed if the list of first stage valves of the GENERAL section contains one associated with this type of operation), the parameters to be set are:
- the first stage control algorithm, that determines the rule used by the Gateway to control the functional block associated with the heating valve; the options available will depend on the type of valve connected (on/off or % value) and the possible presence of the three fancoil speed functional blocks.

  a. *two ON - OFF points (visible with an on/off functional block)*
     The algorithm used for controlling the temperature adjustment system is the classic type, called "2-point control". This type of control involves the turning on and off of the temperature adjustment system following a hysteresis cycle. This means there isn't a single threshold that discriminates between the turning on and off of the system, but two.
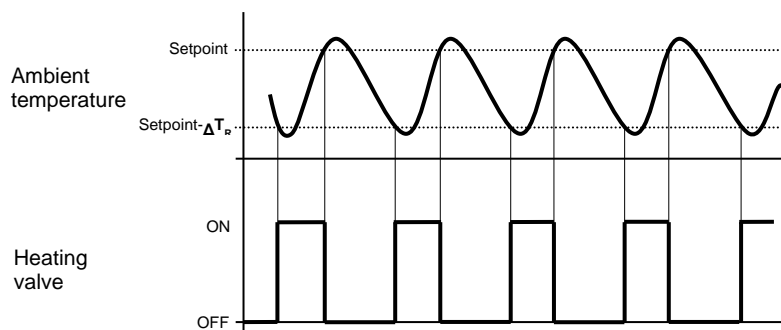
When the measured temperature is lower than the value "setpoint-ΔT_R "(where ΔT_R identifies the value of the heating regulation differential), the device activates the heating system, sending the relative command to the functional block that manages it; when the measured temperature reaches the fixed setpoint value, the device deactivates the heating system, sending the relative command to the functional block.

  b. *proportional integral PI (visible with an on/off or % value functional block)*
     The algorithm used to control the temperature adjustment system allows you to drastically reduce the thermal inertia times introduced by the 2-point control, called proportional integral (PI) control. This type of control involves the modulation of the temperature adjustment system power (duty cycle of the ON-OFF command, or % value) on the basis of the difference between the fixed setpoint and the temperature measured. Two components are needed to calculate the output function: the proportional component and the integral component.

$$u(t) = K_p e(t) + K_i \int_0^t e(\tau)\, d\tau$$

Proportional component

In the proportional component, the output function is proportional to the error (difference between setpoint and measured temperature).

$$P_{\text{out}} = K_p\, e(t)$$

Once the proportional band has been defined, the system power within the band varies between 0% and 100%; outside the band, the power will be maximum or minimum depending on the reference limits.

The width of the proportional band determines the extent of the response to the error. If the band is too "narrow", the system oscillates as it's more reactive; if the band is too "wide", the control system is slow. The ideal situation is when the proportional band is as narrow as possible without causing oscillations.

The diagram below shows the effect of narrowing the proportional band until the oscillation point of the output function. A "wide" proportional band results as a straight line in the control, but with an initial error between the setpoint and the actually perceptible temperature. As the band becomes narrower, the temperature approaches the reference value (setpoint) until it becomes unstable and starts to oscillate around it.

Integral component

The contribution of the integral period is proportional to the error (difference between the setpoint and the measured temperature) and its duration. The integral is the sum of the instantaneous error for every moment of time and provides the accumulated offset that should have been previously corrected. The accumulated error is then added to the regulator output.

$$I_{\text{out}} = K_i \int_0^t e(\tau)\,d\tau$$

The integral period accelerates the dynamics of the process towards the setpoint and eliminates the residuals of the stationary error status that takes place with a pure proportional controller.

The integration time is the parameter that determines the action of the integral component. The longer the integration time, the slower the modification of the output and hence the slower the system response. If the time is too short, the threshold value will be exceeded (overshoot), and the function will swing around the set-point.

The following example shows how the algorithm works with an ON-OFF valve and a percentage valve:

The device continuously adjusts the heating system, modulating the power on the basis of the value calculated by the control algorithm.

If the valve is ON-OFF, the device modulates the system on-off times with a duty-cycle (shown on the right, along the vertical axis) that depends on the output function value calculated at every time interval equal to the cycle time. If the valve is of the percentage type, the device continuously adjusts the heating system, sending the solenoid valve percentage activation values (shown on the vertical axis) that depend on the output function of the algorithm.

With this type of algorithm, there is no longer a hysteresis cycle on the heating device, so the inertia times (system heating and air cooling times) introduced by the 2-point control are eliminated. This produces energy savings because the system does not remain switched on when it is not needed and, once the required temperature has been reached, it continues to provide a heat limited contribution to compensate for the environmental heat dispersion.

c. *3-speed ON-OFF fancoil (visible if the three functional blocks are present in the list of fancoil speeds)*

The type of control that is applied when the fancoil control is enabled is similar to the 2-point control analysed in previous sections, which is to turn the fancoil speed on /off based on the difference between the setpoint that was set and the measured temperature.

The basic difference compared with the 2-point algorithm is that, in this case, there isn't just one stage in which the hysteresis cycle is performed to fix the speed on/off thresholds; there are three. In short, this means that each stage corresponds to a speed and when the difference between the measured temperature and the setpoint that was set causes a certain speed to be

turned on, this means that before turning on the new speed, the other two must absolutely be turned off.

HEATING

The figure refers to the control of the three fancoil speeds for heating. The chart shows that each speed has a hysteresis cycle, and each speed is associated with two thresholds that determine its activation and deactivation. The thresholds are determined by values set for the various regulation differentials, and can be summarised as follows:

- Speed V1: this speed is activated when the temperature is lower than the value "setpoint-$_\Delta T_{valv}$-$_\Delta T_{1\,heat}$" and deactivated when the temperature reaches the value "setpoint-$_\Delta T_{valv}$" (or the "setpoint" value if $_\Delta T_{1\,heat}$=0). The first speed is also turned off when a higher speed needs to be activated

- Speed V2: this speed is activated when the temperature is lower than the value "setpoint-$_\Delta T_{valv}$-$_\Delta T_{1\,heat}$-$_\Delta T_{2\,heat}$" and deactivated when the temperature reaches the value "setpoint-$_\Delta T_{valv}$-$_\Delta T_{1\,heat}$". The second speed is also turned off when the V3 speed needs to be activated

- Speed V3: this speed is activated when the temperature is lower than the value "setpoint-$_\Delta T_{valv}$-$_\Delta T_{1heat}$-$_\Delta T_{2heat}$-$_\Delta T_{3heat}$" and deactivated when the temperature reaches the value "setpoint-$_\Delta T_{valv}$-$_\Delta T_{1heat}$-$_\Delta T_{2heat}$".

With regards the heating solenoid valve, once the measured temperature is lower than the value "setpoint-ΔTvalv", the thermostat sends the activation command to the solenoid valve that manages the heating system; the solenoid valve is deactivated when the detected temperature reaches the fixed setpoint value. In this way, the heating of the fancoil can also be exploited for irradiation, without any speed being activated.

- value of the valve regulation differential (mentioned above). Possible values: from 0.1°C to 2°C.
  This parameter can only be edited if the control algorithm selected is "two points ON - OFF" or "3-speed fancoil ON-OFF".
- width of the proportional band (mentioned above). Possible values: from 1°C to 10°C.
  This parameter can only be edited if the control algorithm selected is "proportional integral PI"
- contribution of the integral action in the proportional integral control, in the form of an integration time. Possible values: from 1 to 240 minutes.
  This parameter can only be edited if the control algorithm selected is "proportional integral PI"
- value of the period within which the device performs the PWM modulation, modifying the duty cycle of the proportional integral PWM (with on-off valve). Possible values: from 5 to 60 minutes.
  This parameter can only be edited if the control algorithm selected is "proportional integral PI" and the valve associated with the first heating stage is of the on-off type.
- minimum variation of the system power value, calculated by the algorithm, for generating a new command to the proportional valve. Possible values: from 1% to 20%.

This parameter can only be edited if the control algorithm selected is "proportional integral PI" and the valve associated with the first heating stage is of the proportional (% value) type.

- value of the regulation differential for fancoil speed 1. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 1 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
- value of the regulation differential for fancoil speed 2. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 2 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
- value of the regulation differential for fancoil speed 3. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 3 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
- limit threshold for the intervention of the supplementary heating system (i.e. second stage). Possible values: from 1°C to 10°C.
  This parameter is only visible, and can only be edited, if the list of second stage valves includes a functional block associated with heating.
  Some heating systems (for example, floor heating) have very high thermal inertia levels and require a considerable amount of time to bring the room temperature into line with the required setpoint. In order to reduce this inertia, another heating system with less inertia is often installed to help the main system to heat the room when the difference between the setpoint and the measured temperature is particularly large. This system, known as 2nd stage, helps to heat the room during the initial phase, then it stops working when the difference between the setpoint and the temperature can be managed faster by the main system (1st stage).
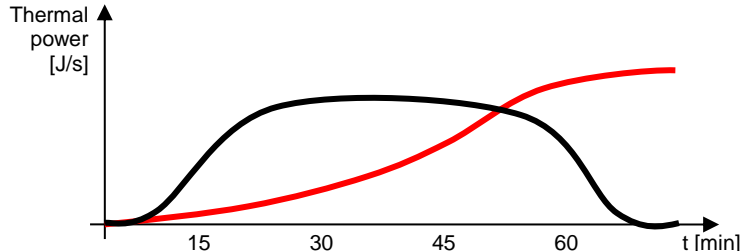
The control algorithm of the second stage is two points ON-OFF, and the intervention thresholds are as follows:
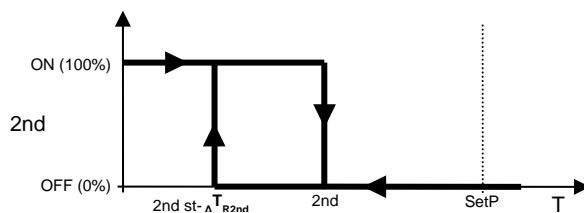
When the measured temperature is lower than the value "2nd st- ΔTR2nd st" (where ΔTR2nd st identifies the value of the regulation differential of 2nd stage heating), the device activates 2nd stage heating by sending the relative command to the valve that manages it. When the measured temperature reaches the value "2nd st" (defined by the setpoint intervention limit 2nd stage), the device deactivates 2nd stage heating.

This makes it clear that there are two decision thresholds for activating and deactivating the 2nd heating stage, the first consists of the value "2nd st- ΔTR2nd st" below which the device turns on the system, and the second consists of the value "2nd st" above which the device switches off the system.

- value of the second stage valve regulation differential (mentioned above). Possible values: from 0.1°C to 2°C.

In the COOLING section (that can be accessed if the list of first stage valves of the GENERAL section contains one associated with this type of operation), the parameters to be set are:
- the first stage control algorithm, that determines the rule used by the Gateway to control the functional block associated with the cooling valve; the options available will depend on the type of valve connected (on/off or % value) and the possible presence of the three fancoil speed functional blocks.
    a. *two ON - OFF points (visible with an on/off functional block)*
       The algorithm used for controlling the temperature adjustment system is the classic type, called "2-point control". This type of control involves the turning on and off of the temperature adjustment system following a hysteresis cycle. This means there isn't a single threshold that discriminates between the turning on and off of the system, but two.
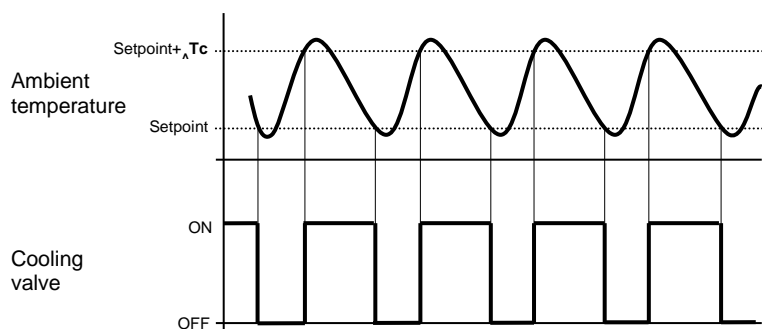
       When the measured temperature is higher than the value "setpoint+$_\Delta$Tc" (where $_\Delta$Tc identifies the value of the cooling regulation differential), the device activates the cooling system by sending the relative command to the functional block that manages it; when the detected temperature reaches the fixed setpoint value, the device deactivates the cooling system.
    b. *proportional integral PI (visible with an on/off or % value functional block)*
       The algorithm used to control the temperature adjustment system allows you to drastically reduce the thermal inertia times introduced by the 2-point control, called proportional integral (PI) control. This type of control involves the modulation of the temperature adjustment system power (duty cycle of the ON-OFF command, or % value) on the basis of the difference between the fixed setpoint and the temperature measured. Two components are needed to calculate the output function: the proportional component and the integral component.

$$u(t) = K_p e(t) + K_i \int_0^t e(\tau)\, d\tau$$

Proportional component
       In the proportional component, the output function is proportional to the error (difference between setpoint and measured temperature).

$$P_{\text{out}} = K_p\, e(t)$$

       Once the proportional band has been defined, the system power within the band varies between 0% and 100%; outside the band, the power will be maximum or minimum depending on the reference limits.
       The width of the proportional band determines the extent of the response to the error. If the band is too "narrow", the system oscillates as it's more reactive; if the band is too "wide", the control system is slow. The ideal situation is when the proportional band is as narrow as possible without causing oscillations.
       The diagram below shows the effect of narrowing the proportional band until the oscillation point of the output function. A "wide" proportional band results as a straight line in the control, but with an initial error between the setpoint and the actually perceptible temperature. As the band becomes narrower, the temperature approaches the reference value (setpoint) until it becomes unstable and starts to oscillate around it.

Output | Proportional band wide | narrow | Temperature
100% | 50% | 0% | tempera ture

Temperature | tempera ture | Proportional band narrowing | Time

<u>Integral component</u>

The contribution of the integral period is proportional to the error (difference between the setpoint and the measured temperature) and its duration. The integral is the sum of the instantaneous error for every moment of time and provides the accumulated offset that should have been previously corrected. The accumulated error is then added to the regulator output.

$$I_{\text{out}} = K_i \int_0^t e(\tau)\, d\tau$$

The integral period accelerates the dynamics of the process towards the setpoint and eliminates the residuals of the stationary error status that takes place with a pure proportional controller.

The integration time is the parameter that determines the action of the integral component. The longer the integration time, the slower the modification of the output and hence the slower the system response. If the time is too short, the threshold value will be exceeded (overshoot), and the function will swing around the set-point.

The following example shows how the algorithm works with an ON-OFF valve and a percentage valve:

Algorithm output

Cooling valve ON-OFF (PWM)

Algorithm output

Cooling valve % value (continuous)

The device continuously adjusts the cooling system, modulating the power on the basis of the value calculated by the control algorithm.

If the valve is ON-OFF, the device modulates the system on-off times with a duty-cycle (shown on the right, along the vertical axis) that depends on the output function value calculated at every time interval equal to the cycle time. If the valve is of the percentage type, the device continuously adjusts the heating system, sending the solenoid valve percentage activation values (shown on the vertical axis) that depend on the output function of the algorithm.

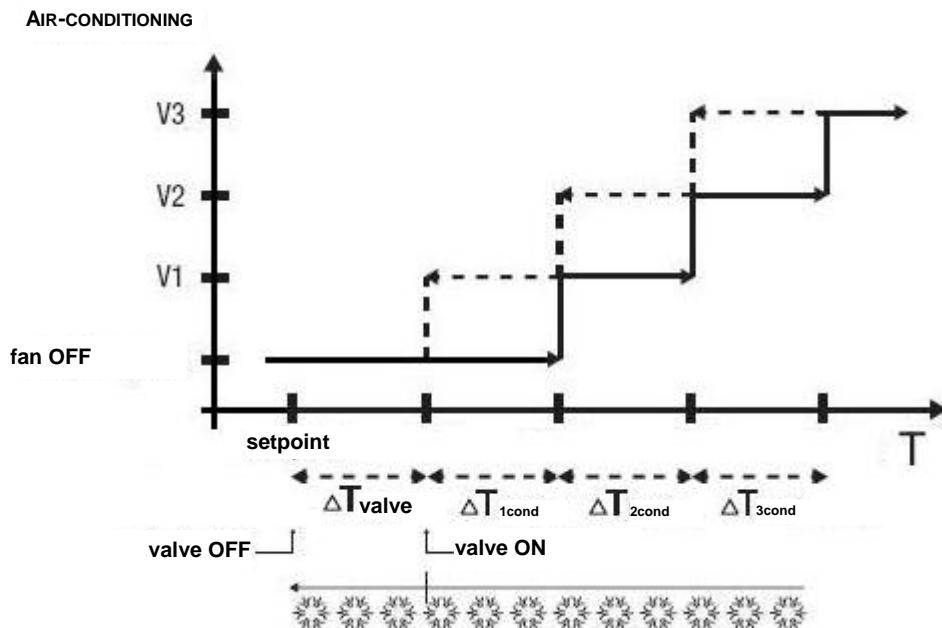With this type of algorithm, there is no longer a hysteresis cycle on the cooling device, so the inertia times (system cooling and heating times) introduced by the 2-point control are eliminated. This produces energy savings because the system does not remain switched on when it is not needed and, once the required temperature has been reached, it continues to provide a limited contribution of cold air to compensate for the contribution of environmental heat.

c. *3-speed ON-OFF fancoil (visible if the three functional blocks are present in the list of fancoil speeds)*

The type of control that is applied when the fancoil control is enabled is similar to the 2-point control analysed in previous sections, which is to turn the fancoil speed on /off based on the difference between the setpoint that was set and the measured temperature.

The basic difference compared with the 2-point algorithm is that, in this case, there isn't just one stage in which the hysteresis cycle is performed to fix the speed on/off thresholds; there are three. In short, this means that each stage corresponds to a speed and when the difference between the measured temperature and the setpoint that was set causes a certain speed to be turned on, this means that before turning on the new speed, the other two must absolutely be turned off.

AIR-CONDITIONING

The figure refers to the control of the fancoil speeds for cooling. The chart shows that each speed has a hysteresis cycle, and each speed is associated with two thresholds that determine its activation and deactivation. The thresholds are determined by values set for the various regulation differentials, and can be summarised as follows:

- Speed V1: this speed is activated when the temperature is higher than the value "setpoint+$_\Delta$T$_{valv}$+$_\Delta$T$_{1cond}$" and deactivated when the temperature reaches the value "setpoint+$_\Delta$T$_{valv}$" (or the "setpoint" value if $_\Delta$T$_{1\ cond}$=0). The first speed is also turned off when a higher speed needs to be activated

- Speed V2: the speed is turned on when the temperature value is higher than the value "setpoint+$_\Delta$T$_{valv}$+$_\Delta$T$_{1\ cond}$+$_\Delta$T$_{2\ cond}$" and turned off when the temperature value reaches the value "setpoint+$_\Delta$T$_{valv}$+$_\Delta$T$_{1\ cond}$". The second speed is also turned off when the V3 speed needs to be activated

- Speed V3: the speed is turned on when the temperature value is higher than the value "setpoint+$_\Delta$T$_{valv}$+$_\Delta$T$_{1\ cond}$+$_\Delta$T$_{2\ cond}$+$_\Delta$T$_{3\ cond}$" and turned off when the temperature value reaches the value "setpoint+$_\Delta$T$_{valv}$+$_\Delta$T$_{1\ cond}$+$_\Delta$T$_{2\ cond}$"

With regards the air cooling solenoid valve, once the measured temperature is higher than the value "setpoint+$_\Delta$T$_{valv}$", the thermostat sends the activation command to the solenoid valve that manages the air cooling system; the solenoid valve is deactivated when the detected temperature reaches the fixed setpoint value. In this way, the air cooling of the fancoil can also be exploited for irradiation, without any speed being activated.

- value of the valve regulation differential (mentioned above). Possible values: from 0.1°C to 2°C. This parameter can only be edited if the control algorithm selected is "two points ON - OFF" or "3-speed fancoil ON-OFF".

- width of the proportional band (mentioned above). Possible values: from 1°C to 10°C. This parameter can only be edited if the control algorithm selected is "proportional integral PI"

- contribution of the integral action in the proportional integral control, in the form of an integration time. Possible values: from 1 to 240 minutes. This parameter can only be edited if the control algorithm selected is "proportional integral PI"

- value of the period within which the device performs the PWM modulation, modifying the duty cycle of the proportional integral PWM (with on-off valve). Possible values: from 5 to 60 minutes. This parameter can only be edited if the control algorithm selected is "proportional integral PI" and the valve associated with the first heating stage is of the on-off type.

- minimum variation of the system power value, calculated by the algorithm, for generating a new command to the proportional valve. Possible values: from 1% to 20%. This parameter can only be edited if the control algorithm selected is "proportional integral PI" and the valve associated with the first heating stage is of the proportional (% value) type.

- value of the regulation differential for fancoil speed 1. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 1 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
- value of the regulation differential for fancoil speed 2. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 2 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
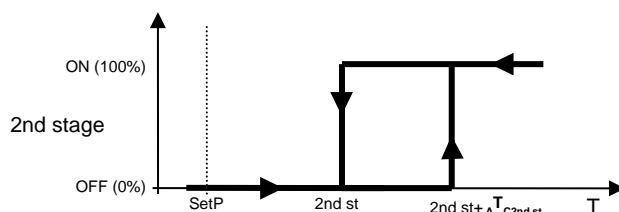- value of the regulation differential for fancoil speed 3. Possible values: from 0.1°C to 2°C.
  This parameter is only visible, and can only be edited, if the control algorithm selected is "3-speed fancoil ON-OFF"
- delay between the deactivation of fancoil speed 3 and the activation of the new speed calculated by the algorithm. Possible values: from 0 to 30 seconds.
- limit threshold for the intervention of the supplementary cooling system (i.e. second stage). Possible values: from 1°C to 10°C.
  This parameter is only visible, and can only be edited, if the list of second stage valves includes a functional block associated with cooling.
  Some cooling systems have very high inertia levels and take a long time to bring the room temperature into line with the required setpoint. In order to reduce this inertia, another cooling system with less inertia is often installed to help the main system to heat the room when the difference between the setpoint and the measured temperature is particularly large. This system, known as 2nd stage, helps to cool the room during the initial phase, then it stops working when the difference between the setpoint and the temperature can be managed faster by the main system (1st stage).
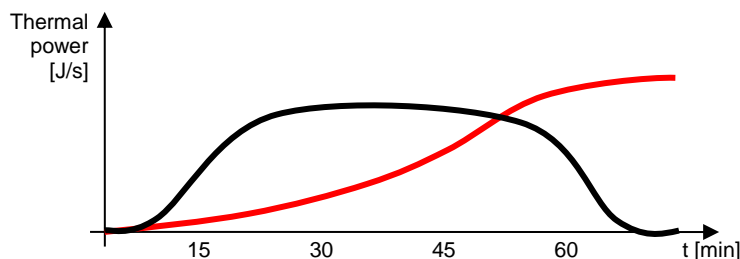
The control algorithm of the second stage is two points ON-OFF, and the intervention thresholds are as follows:

When the measured temperature is higher than the value "2nd st+ $_\Delta T_{C2nd\ st}$" (where $_\Delta T_{C2nd\ st}$ identifies the value of the regulation differential of 2nd stage cooling), the device activates 2nd stage cooling by sending the relative command to the valve that manages it. When the measured temperature reaches the value "2nd st" (defined by the setpoint intervention limit 2nd stage), the device deactivates 2nd stage cooling.

This diagram clearly shows that there are two decision thresholds for activating and deactivating 2nd stage cooling. The first is the value "2nd st+ $_\Delta T_{C2nd\ st}$", above which the device activates the system, and the second is the value "2nd st", below which the device deactivates the system.

- value of the second stage valve regulation differential (mentioned above). Possible values: from 0.1°C to 2°C.

In the SETPOINT section, the parameters to be set are:
- The value of the **comfort** mode setpoint for **heating**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
  ATTENTION: when setting this value, make sure it's higher than the value set in the "Pre-comfort setpoint" parameter for heating.
- The value of the **pre-comfort** mode setpoint for **heating**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
  ATTENTION: when setting this value, make sure it's higher than the value set in the "Economy setpoint" parameter for heating.
- The value of the **economy** mode setpoint for **heating**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
- The value of the **off (antifreeze)** mode setpoint for **heating**. Possible values: from 2°C to 10°C.
- The value of the **comfort** mode setpoint for **cooling**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
  ATTENTION: when setting this value, make sure it's lower than the value set in the "Pre-comfort setpoint" parameter for cooling.
- The value of the **pre-comfort** mode setpoint for **cooling**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
  ATTENTION: when setting this value, make sure it's lower than the value set in the "Economy setpoint" parameter for cooling.
- The value of the **economy** mode setpoint for **cooling**. Possible values: from 10°C to 35°C.
  This parameter is only visible, and can only be edited, if the type of timed thermostat control set in the GENERAL section is HVAC mode.
- The value of the **off (high temperature protection)** mode setpoint for **cooling**. Possible values: from 35°C to 40°C.

In the RELATIVE HUMIDITY section, the parameters to be set are:
- Selection of the relative humidity sensor that will provide the value to be monitored; press the ✎ icon to see the pop-up with a list of the functional blocks compatible with the function (and not already used in the other timed thermostats created). Once the sensor has been selected, all the other parameters (see below) will appear. Press the ✖ icon to delete the functional block associated with the humidity sensor.
- Static correction of the relative humidity value received from the humidity sensor (if it's influenced by other factors). Possible values: from -20% to +20%
- Selection (optional) of the possible functional block that manages room humidification; press the ✎ icon to see the pop-up with a list of the functional blocks compatible with the function (and not already used in the other timed thermostats created). Press the ✖ icon to delete the functional block associated with the humidifier.

The humidification algorithm works on the basis of an ON-OFF hysteresis:

ON

humidification

OFF

Limit threshold - hysteresis   Limit threshold   RH[%]
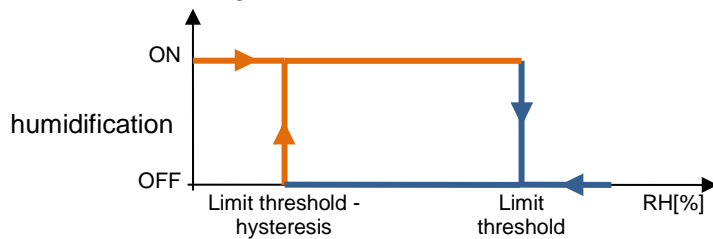
When the relative humidity measured by the sensor is lower than the "Limit threshold - hysteresis" value, the device activates the humidification system; when the reference relative humidity level reaches the limit threshold value, the device deactivates the system.

- Value of the limit threshold associated with humidification. Possible values: from 0% to 100%.
  This parameter can only be edited if a functional block has been selected for the humidification phase.
- Value of the hysteresis that, when subtracted from the limit threshold, helps to define the deactivation threshold of the humidification system. Possible values: from 1% to 20%.
  This parameter can only be edited if a functional block has been selected for the humidification phase.
- Selection (optional) of the possible functional block that manages room dehumidification; press the ✏ icon to see the pop-up with a list of the functional blocks compatible with the function (and not already used in the other timed thermostats created). Press the ✖ icon to delete the functional block associated with the dehumidifier.
  The dehumidification algorithm works on the basis of an ON-OFF hysteresis:

ON

dehumidification

OFF

Limit threshold   Limit threshold+hysteresis   RH[%]
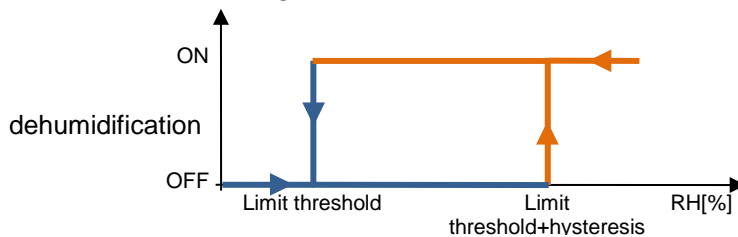
When the relative humidity measured by the sensor is higher than the "Limit threshold + hysteresis" value, the device activates the dehumidification system; when the reference relative humidity level reaches the limit threshold value, the device deactivates the system.

- Value of the limit threshold associated with dehumidification. Possible values: from 0% to 100%.
  This parameter can only be edited if a functional block has been selected for the dehumidification phase.
- Value of the hysteresis that, when added to the limit threshold, helps to define the activation threshold of the dehumidification system. Possible values: from 1% to 20%.

Press the "OK" push-button to see the weekly programming of the temperature adjustment profile associated with the timed thermostat, heating and/or cooling (depending on the controls enabled).

Underneath the name of the timed thermostat, there are two push-buttons for selecting the profile to be programmed: heating or cooling; (for each timed thermostat, in fact, there are two distinct profiles for the two types of operation). If one type of operation hasn't been enabled, the relative push-button will be disabled.

The central part of the page will show the weekly programming of the timed thermostat.
At the top there's a time line (00-24). To alter the time resolution, bring the cursor inside the programming grid and rotate the mouse wheel upwards (to increase the resolution) or downwards (to reduce it). When the resolution doesn't allow you to see the entire 00-24 scale, you can move along the time axis by selecting any point on it and dragging it to the right or left.

Each row represents one day of the week.

To add a new time band in a specific day, press the **+** icon on the right of the name of the day concerned. In the pop-up menu that appears, select the band start time and the threshold value (HVAC or setpoint mode, depending on the type of timed thermostat); the band end time coincides with the start time of the next band, or with the end of the day.

To repeat the programming of one day on another day, press the icon to the right of the day whose programming you want to repeat; from the pop-up menu that appears, select the day on which you want to copy the programming, then confirm.

To enable/disable the programming of a day of the week, press the coloured band to the left of the name:

  Programming enabled

  Programming disabled

On the time line for each day, there are the time bands with the start time and relative value. Daily programming can never be blank - there must be at least one band (the band that begins at 00:00 can't be deleted).

Inside the band, you can see:

- the value (HVAC or setpoint mode) active in that time band
- the band start time (underneath the value)
- ✏ push-button for modifying the start time and the value
- 🗑 push-button for deleting the band

If the profile is "heating", the various time bands are coloured in different shades of red according to their value; the same for the "cooling" profile, but in blue.

The start and end times of a band can be modified directly from the time grid: select the required band (it will turn yellow), then select the left-hand edge of the band and drag it to the right or left to change the start time in 15-minute steps, or select and drag the right-hand edge to change the end time. Release to save the modification.

NOTE: the implementation of the profile associated with a specific time period (calendar) isn't defined in programming; it's one of the profile activation options (disabled, enabled, enabled with calendar) managed via the app. Once activated, the associated profile will be activated/visualised depending on whether the timed thermostat is in heating or cooling mode (without the user having to select it).

When the central area shows the weekly programming of the timed thermostat, a new icon ⚙ will appear next to the name of the timed thermostat, along with the validity status of the element.

⚠ ✅ Icon indicating the validity status of the timed thermostat: orange if it's incomplete and can't therefore be used in the project, green if it's complete and can be used. If the timed thermostat is incomplete, click on the orange icon to see the type of error.

⚙ Press this icon to see the parameters that determine the behaviour of the timed thermostat (described above).

ATTENTION: an incomplete timed thermostat isn't saved on the cloud. This means that it will no longer be present when the page is changed, or when the project is closed and reopened, even if the configuration has been sent to the area shared with clients (by pressing the 🔘 push-button).

Every time you select a timed thermostat from the TIMED THERMOSTATS column, the weekly programming of the timed thermostat profile will be visualised; to alter the parameters that determine the behaviour of the timed thermostat, press the ⚙ icon next to the name of the group.

All the timed thermostats created are listed in alphabetical order in the TIMED THERMOSTATS column (on the right).

CRONOTHERMOSTATS	+	🗑
Search		
Chronothermostat 1		
Chronothermostat 2		
Chronothermostat 3		
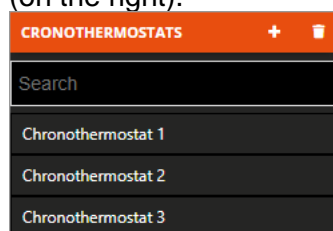
To remove a timed thermostat from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the timed thermostats simultaneously by pressing the 🗑 icon in the right-hand column ("TIMED THERMOSTATS").

ATTENTION: a local timed thermostat can only be removed if it isn't used in other functions (apart from supervision); otherwise, an error message will appear and the profile won't be deleted.
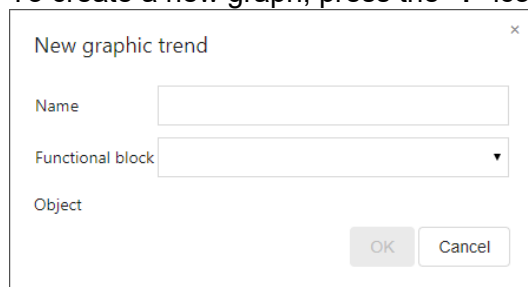
Graphic trends

A graphic trend shows the values of specific variables (statuses or analogue sizes) in a time diagram, with daily/monthly/yearly time resolution, and offers the possibility to make comparisons with previous periods.

The Smart Gateway sends the status variations of all the field variables in the system, on the basis of predefined rules. When a graphic trend is configured, the app is instructed to download the data from the cloud and to show the values in a chart.

Once you have selected the "Functions" menu on the first row and the "GRAPHIC TRENDS" menu on the second, the GRAPHIC TRENDS column will appear (on the right).
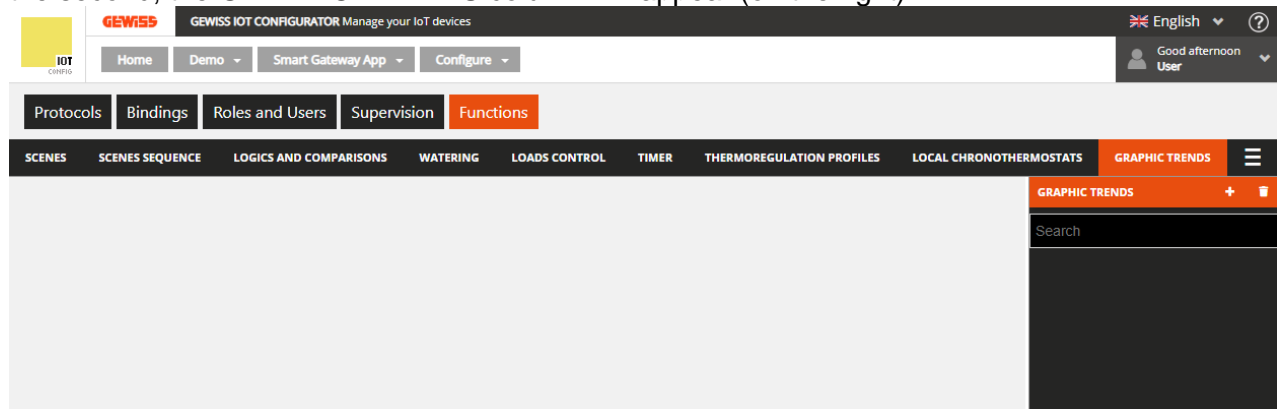
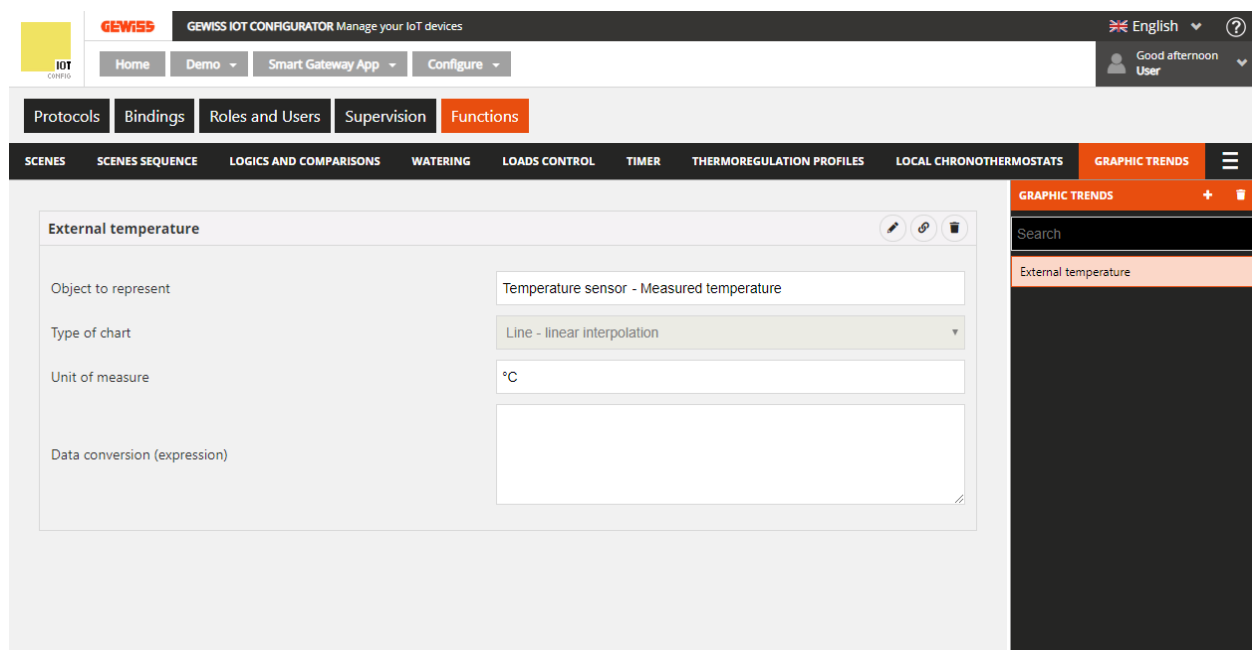To create a new graph, press the "**+**" icon in the right-hand column ("GRAPHIC TRENDS").

In the pop-up menu that appears, enter the name and
- select the functional block of the size for which you want to show the graph.
- select the variable (of the selected functional block) for which you want to show the trend over time; the options will depend on the functional block you've selected

Next to the name of the graphic trend, you can see:

✎ **Rename** push-button for modifying the name of the graphic trend

🔗 **View connections** push-button for displaying all the pages where the graphic trend is used (select the required page to gain direct access)

🗑 **Delete** push-button for eliminating the graphic trend

The central part of the page will show the parameters that determine the type of graph you want to use to show the value.
The parameters to be set are:
- the type of graph (the options that are visible will depend on the size selected).
- the measurement unit of the size (X axis) shown on the graph (pre-set automatically on the basis of the size selected)

All the trends created are listed in alphabetical order in the GRAPHIC TRENDS column (on the right).
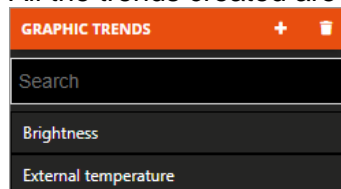
To remove a trend from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the graphic trends simultaneously by pressing the 🗑 icon in the right-hand column ("GRAPHIC TRENDS").

IP cameras

The app is able to visualise the video flow - in high and low resolution - of IP cameras that can be reached via an http or rtsp address, or that support the ONVIF communication protocol.
In addition, the following commands are available (only for the ONVIF camera models that support them):
- PTZ regulation (Pan, Tilting, Zoom)
- image regulation (light intensity, contrast, colour)
- IR filter setting (night-time or daytime mode)
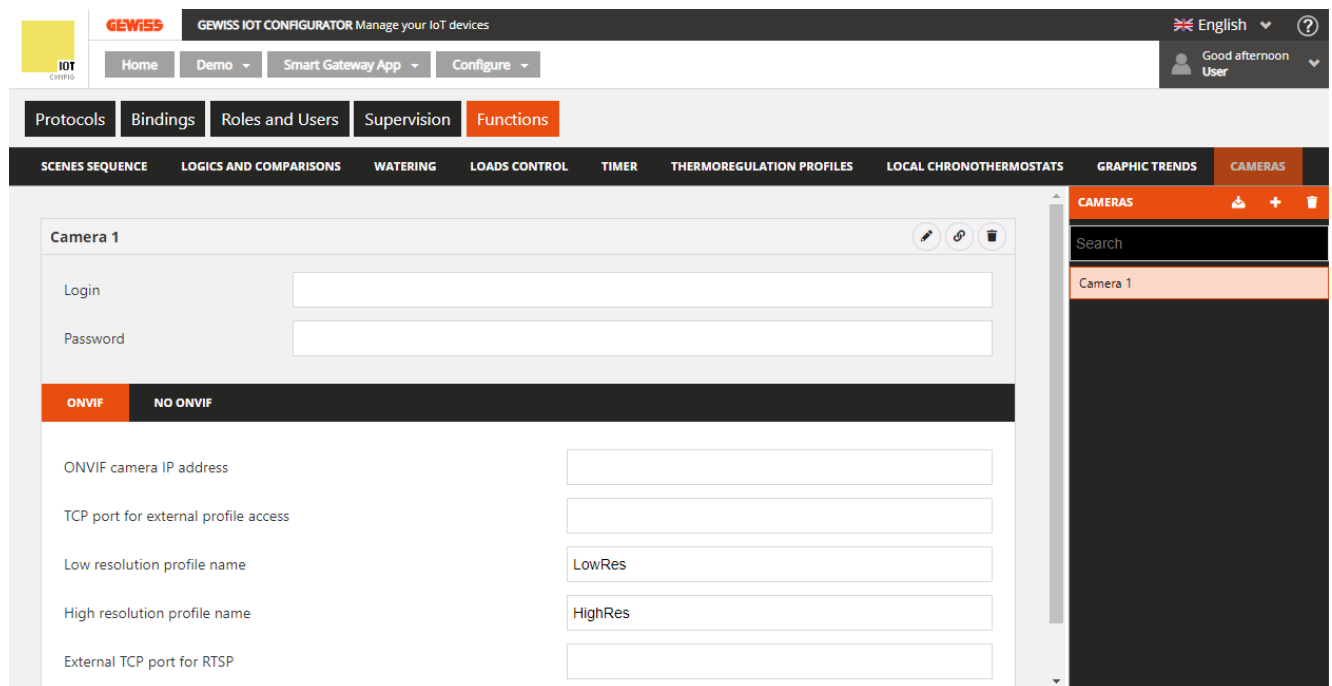- activation/deactivation of the local video recording function of the camera
Unlike the procedure for other functions, video flow transmission isn't via the cloud: the connection with the camera is point-point.
A camera can be added to the system by directly entering the parameters, or by importing the configuration made via the dedicated **IP CAMERAS CONFIGURATOR** software.
Once you have selected the "Functions" menu on the first row and the "CAMERAS" menu on the second, the CAMERAS column will appear (on the right).
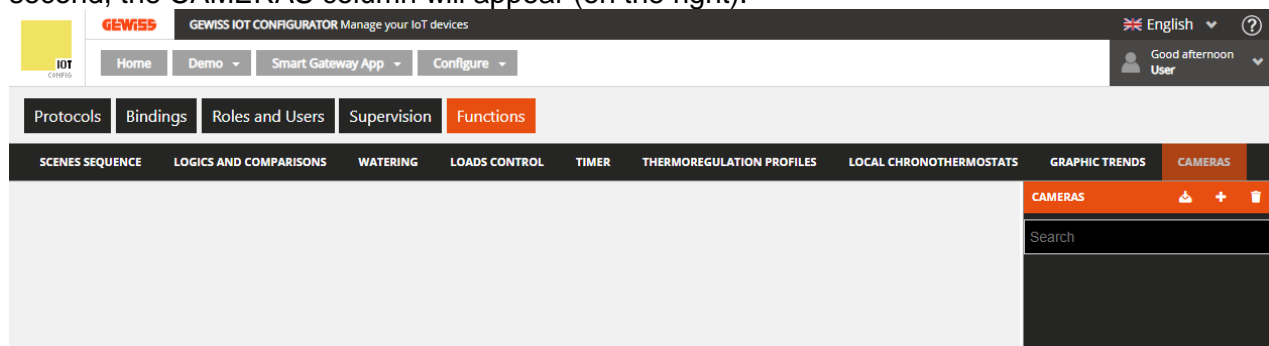
To create a new camera, press the "**+**" icon in the right-hand column ("CAMERAS").
Enter the name you want to associate with the camera, then confirm.

Next to the name of the camera you will see:

✏️ **Rename** push-button for modifying the name of the camera

🔗 **View connections** push-button for displaying all the pages where the camera is used (select the required page to gain direct access)

🗑️ **Delete** push-button for eliminating the camera

The central part of the page will show the parameters for the local and remote connection to the camera. The parameters to be set are:
- Login and Password needed to authenticate the camera (if required).
  NOTE: the access credentials requested are often (but not always) the same ones used to access camera configuration via the web.
- Select the type of camera used - ONVIF if the IP camera in question implements the ONVIF communication protocol, or NON ONVIF if it doesn't implement it. The configuration parameters will change according to the type selected.

In the ONVIF section, the parameters to be set are:
- IP address of the camera in the local network it's installed in, and possible access port (if different from the standard one). Example: "address":"port" → **192.168.1.20:80**
- External TCP port defined in the port forwarding rule created on the domestic Internet router (if a VPN connection isn't used) for ONVIF channel; if the port is not specified, the one used for internal communication will be used (usually port 80).
- Name of the profile (implemented by the camera) that you want to use to view the flow in low resolution. Every ONVIF camera implements different profiles with different resolution formats, audio/video source and frame rates, so you can choose the video flow best suited to your own needs. Some cameras also allow you to create new profiles. To complete this field, you need to know the list of profiles available, and the relative name. At least one of the two profiles for low or high resolution must be specified in order to view the camera.

- Name of the profile (implemented by the camera) that you want to use to view the flow in high resolution. At least one of the two profiles for low or high resolution must be specified in order to view the camera.
- External TCP port defined in the port forwarding rule created on the domestic Internet router (if a VPN connection isn't used) for the RTSP or HTTP video flow; if the port isn't specified, the one used for internal communication will be used (usually port 554 for RTSP and port 80 for HTTP).
  NOTE: when controlling several cameras, it's necessary to configure different external ports to avoid any conflict.

In the NON ONVIF section, the parameters to be set are:
- URL address (http or rtsp) of the camera for viewing the video flow in low resolution. At least one of the two addresses for low or high resolution must be specified in order to view the camera.
- URL address (http or rtsp) of the camera for viewing the video flow in high resolution. At least one of the two addresses for low or high resolution must be specified in order to view the camera.
- External TCP port defined in the port forwarding rule created on the domestic Internet router (if a VPN connection isn't used) for the rtsp or http video flow; if the port isn't specified, the one used for internal communication will be used (usually port 554 for RTSP and port 80 for HTTP).
  NOTE: when controlling several cameras, it's necessary to configure different external ports to avoid any conflict.

For the correct configuration of the camera access parameters, you are advised to use the **IP CAMERAS CONFIGURATOR** tool designed to manage the cameras.
The configuration made with the external tool can be imported in the IoT Configurator by pressing the ⬤ **Import camera configuration** icon in the CAMERAS column (on the right).
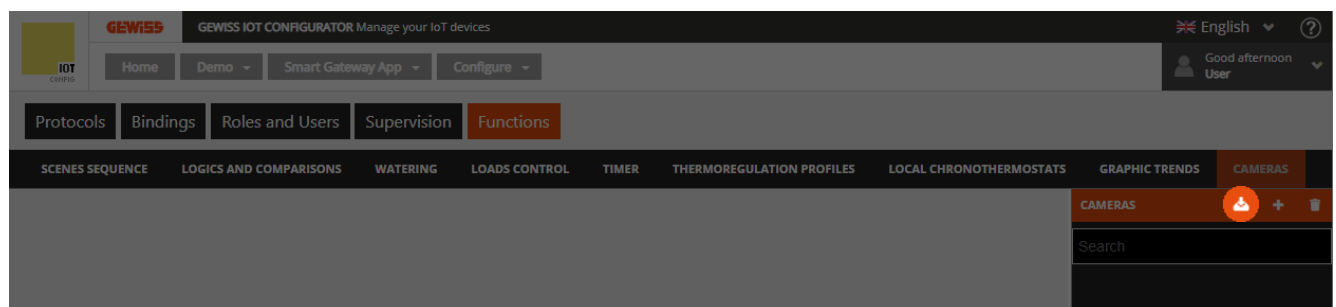
Select the file and then confirm its opening.
The IoT Configurator will begin processing the file and, at the end, you will see all the cameras imported along with the relative parameters.

All the cameras created are listed, in order of creation, in the CAMERAS column (on the right).
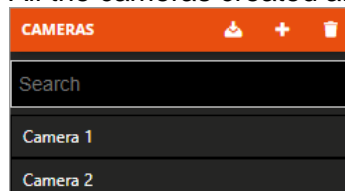
To remove a camera from the list, just activate the context menu of the object and select the "Delete" command. You can even eliminate all the cameras simultaneously by pressing the 🗑 icon in the right-hand column ("CAMERAS").
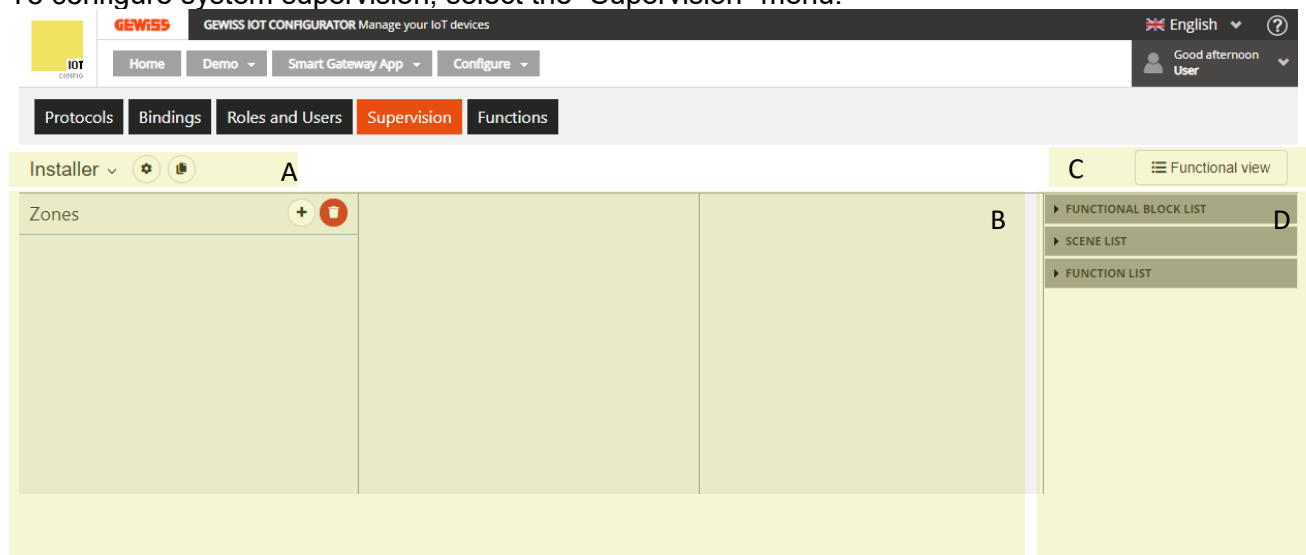
SUPERVISION

Supervision via the app offers a structured way of accessing and interacting with the commands and the visualisation of the statuses of objects or devices (lights, roller shutters, thermostats, etc.) in the system, regardless of the communication protocol used.

There are three types of navigation logic:

- Zones/Environments (both in list format)

  The element/object controlled is represented within the environment in list mode, without any reference to its real position in the system/environment visualised.

- Zones/Environments with the elements in the environment represented on a map/planimetry (tablet version only)

  The elements of each environment can be represented on a planimetry map where the object's position represents its real physical position in the system.

- Functional categories (Lighting, Roller shutters, Temperature adjustment, Energy, etc.)

  The system elements/objects are automatically sub-divided on the basis of their functional category (lighting, roller shutters, temperature adjustment, energy, etc.). When the elements are represented in this way, they can be accessed and commanded in the same way as for Zones/Environments navigation.

To configure system supervision, select the "Supervision" menu.

A. Role for which you are editing the supervision and general settings

In this section, you can select the role for which you want to edit the supervision. Press on the name shown to see a list of all the roles in the project; select the required item to see the navigation tree for the supervision of that role.

Press the ⚙ icon next to the name of the role to see a pop-up with its general supervision settings; in particular, you can enable the consent (for users covering that role) to create certain functions via the app.

Use the 📑 icon to copy the supervision from another role; this greatly speeds up the creation process.

B. Supervision design area

Zones/Environments and Functional navigation is constructed in this area, by defining the elements you want to view/command. It will be described in detail later on.

C. See elements associated with the functional view

This push-button allows you to pass from the programming of Zones/Environments navigation to that of Functional navigation.

The design area will differ according to which type of navigation you are programming.

D. List of the elements that can be used in the design area

This area contains all the elements that can be inserted in the supervision of the current role; to ensure that a specific user associated with a role can access the commands/statuses of the functional blocks or the created functions, you must insert the relative element in the Zones/Environments navigation tree or in the Functional view.

The elements are sub-divided into: Functional blocks (KNX and ZigBee), Scenes, Functions, Cameras (functional only) and graphical trends (functional only). In each sub-section there's a search field for filtering the elements.

The elements highlighted in grey are inserted in the environment or Functional view selected.

Creating the Zones/Environments structure

The design area for creating Zones/Environments navigation consists of three columns:

- list of zones
- list of environments in the selected zone
- list of elements in the selected environment

To add an zone, press the ⊕ push-button next to the name "Zones"; once the name has been inserted, the element that represents the zone will be created.
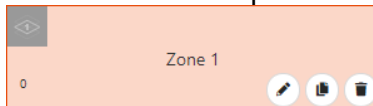

In the middle of the element you can see the name assigned to the zone that will be visualised in the supervision via app.

✏ **Modify** push-button for modifying the name of the zone

▣ **Copy** push-button for creating an identical copy of the zone (including environments and the elements inside them)

🗑 **Delete** push-button for eliminating the zone (and all the environments and the elements inside them)

At the bottom left, you can see the number of environments contained in the zone.

At the top left, there's the icon representing the zone in the supervision via app. If no icon is defined, the default one will appear during configuration; when the push-button is pressed, a pop-up appears where you can select the icon.
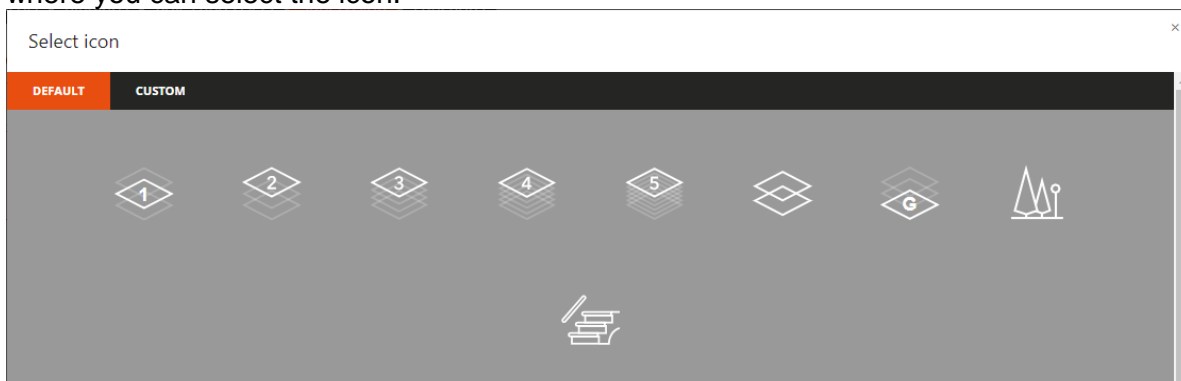
The pop-up is formed of two menus: DEFAULT and CUSTOM.

The first menu contains the icons made available by GEWISS.

The second menu shows all the icons imported in the system.

To import an icon, press the "Import icon" push-button and then select the required icon.
Maximum file size: 1 MB
File extensions supported: svg/png/jpeg

The order in the list of zones determines the order of visualisation in the app (it can be modified directly from the app); to move a zone, just select it and drag it to the required position.

To delete all the zones simultaneously, press the 🔴 icon next to the name "Zones".

There are no limits to the number of zones that can be created.

To add an environment to a zone, select the required element from the "Zones" column and then press the ⊕ push-button next to the name "Environments"; once the name has been inserted, the element that represents the environment will be created.
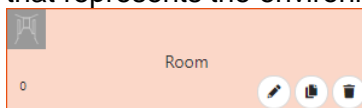
In the middle of the element you can see the name assigned to the environment that will be visualised in the supervision via app.

🖊 **Modify** push-button for modifying the name of the environment

📋 **Copy** push-button for creating an identical copy of the environment (including the elements inside it)

🗑 **Delete** push-button for eliminating the environment (and all the elements inside it)

At the bottom left, you can see the number of elements contained in the environment.

At the top left, there's the icon representing the environment in the supervision via app. If no icon is defined, the default one will appear during configuration; when the push-button is pressed, a pop-up appears where you can select the icon.
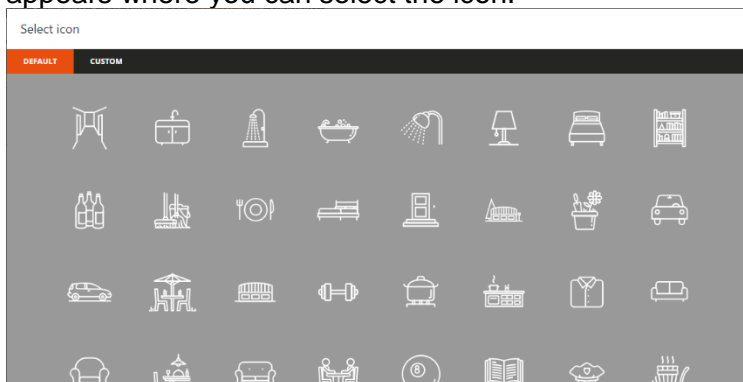
The pop-up is formed of two menus: DEFAULT and CUSTOM.

The first menu contains the icons made available by GEWISS.

The second menu shows all the icons imported in the system.

To import an icon, press the "Import icon" push-button and then select the required icon.
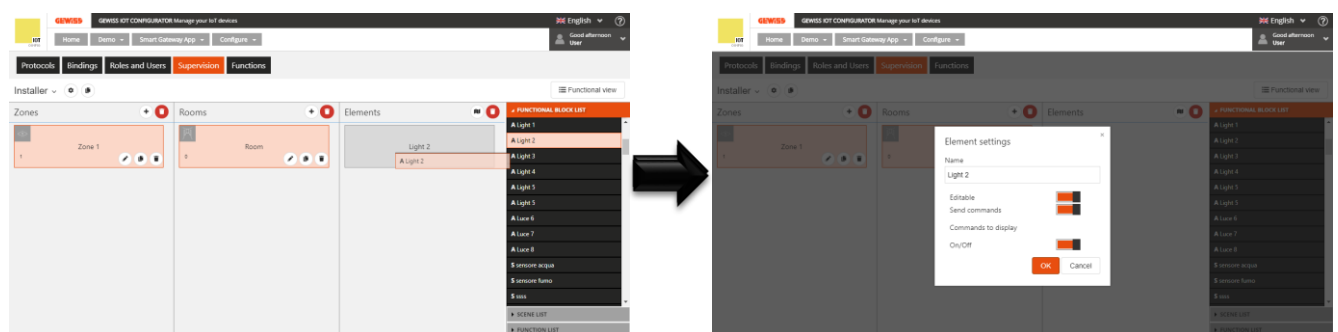Maximum file size: 1 MB
File extensions supported: svg/png/jpeg

The order in the list of zone environments determines the order of visualisation in the app (it can be modified directly from the app); to move an environment, just select it and drag it to the required position.

To delete all the environments of a zone simultaneously, press the 🗑 icon next to the name "Environments".

There are no limits to the number of environments that can be created in a zone.

To add an element (functional block, scene, function) in the environment of a zone, select the required zone from the "Zones" column and the required environment from the "Environments" column, then select the element from the list (right-hand column) and drag it into the supervision design area; release the object in the "Elements" section, in the required visualisation position. At the end of this operation, a pop-up menu will appear where you can select:

- name of the element that will be visualised in the supervision via app (can be different from the name used in configuration)
- enabling of the modification of the element via app: if the **Modifiable** box is ticked, the element can be modified via the app; otherwise, the element will be visible but its configuration can't be altered
- enabling of the sending of element-related commands from the app: if the **Send commands** box is ticked, all the graphic widgets of the element that involve the sending of commands will be enabled to do so; otherwise, the widgets will indicate the status but it will be impossible to send commands.
- Selection of the element commands/statuses that you want to make available on the app; the list of commands available will depend on the type of element you are configuring

Once the settings have been confirmed, the element will be shown in the list of the Elements of the environment, in the position defined when you dragged it, and it will be automatically shown in the Functional view as well.
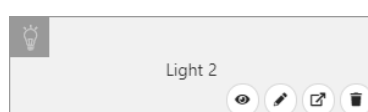

In the middle of the element you can see the name assigned to it and visualised in the supervision via app.

👁 **Go to the object** push-button for passing directly to the page dedicated to the element (for making quick modifications)

Modify push-button for reactivating the pop-up where you can modify the name, enable the modification and sending of commands, and select the commands to be visualised

Delete push-button for eliminating the element (along with the possible copy shown in the Functional view)

ATTENTION: if a function element has been inserted in the supervision of a specific role, but not the object controlled by the same function, you won't be able to modify its configuration via the app even if the modify box has been ticked. Example: if you insert a timer that controls a light, but not the light itself, you can't modify the timer via the app.

At the top left, there's the icon representing the element in the supervision via app. If no icon is defined, the default one will be uploaded in the app and the [+] push-button will appear during configuration; when the push-button is pressed, a pop-up appears where you can select the icon.
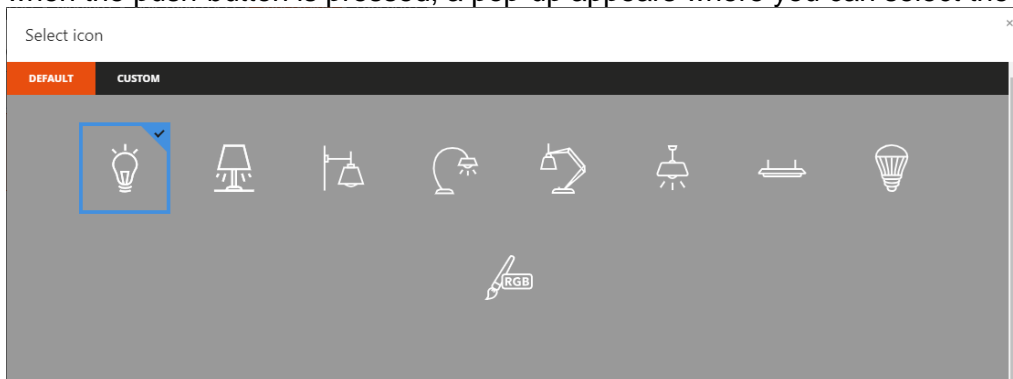
The pop-up is formed of two menus: DEFAULT and CUSTOM.

The first menu contains the icons made available by GEWISS.

The second menu shows all the icons imported in the system.

To import an icon, press the "Import icon" push-button and then select the required icon.
Maximum file size: 1 MB
File extensions supported: svg/png/jpeg

An element can be inserted in several environments within the supervision of a specific role; the first request inserted will be automatically shown in the Functional view as well, speeding up the configuration phase (the element can be removed from the Functional view, if you prefer).

The order in the list of an environment elements determines the order of visualisation in the app (it can be modified directly from the app); to move an element, just select it and drag it to the required position.

To delete all the elements of an environment simultaneously, press the 🗑 icon next to the name "Elements".

Environments with elements represented on a planimetry (Tablet)

In supervision via a tablet, the elements can be viewed in list format, or arranged on an image (planimetry view).

In the IoT Configurator, you can upload the background image for each environment and arrange the elements on the images. In any case, you can still modify the organisation of the elements on the page via the app.

To configure the planimetry view of an environment, select the required zone from the "Zone" column and the required environment from the "Environments" column, then press the 📖 icon in the "Elements" column.
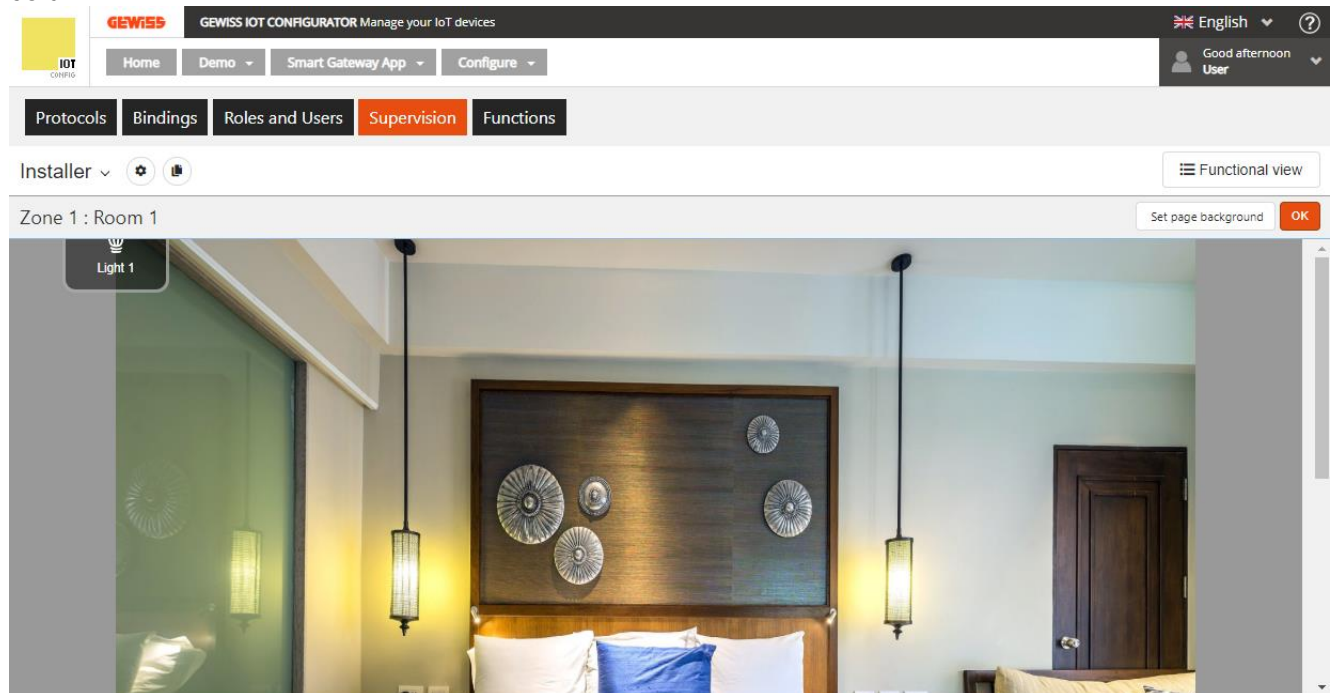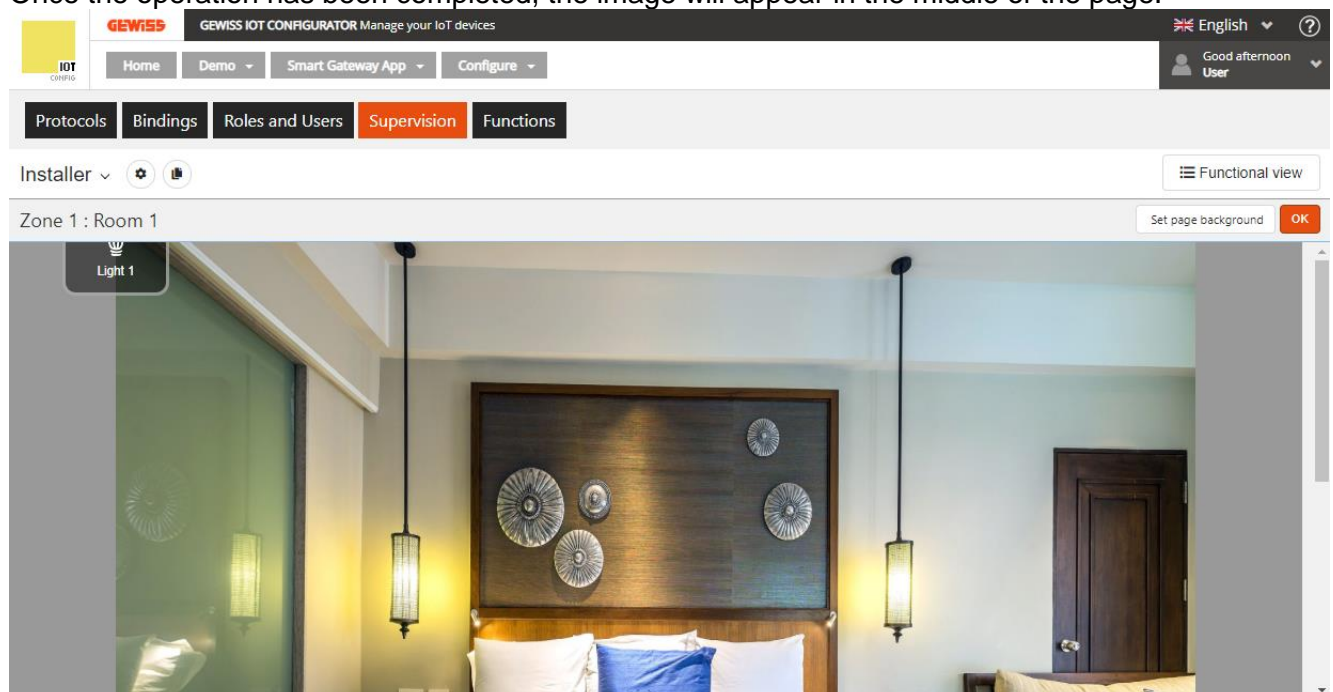

Press the "Set page background" push-button to upload the image you want to associate with the environment.

Maximum file size: 1 MB

File extensions supported: svg/png/jpeg

Once the operation has been completed, the image will appear in the middle of the page.

All the elements in the environment are shown in the top left corner; to move them, select the element and drag it to the required position, then release it.

In the planimetry view of an environment, you can't add or remove elements - you can only rearrange them on the page.

After reorganising the elements, press the OK push-button to return to the list view.

Repeat the operation for all the environments of all the zones.
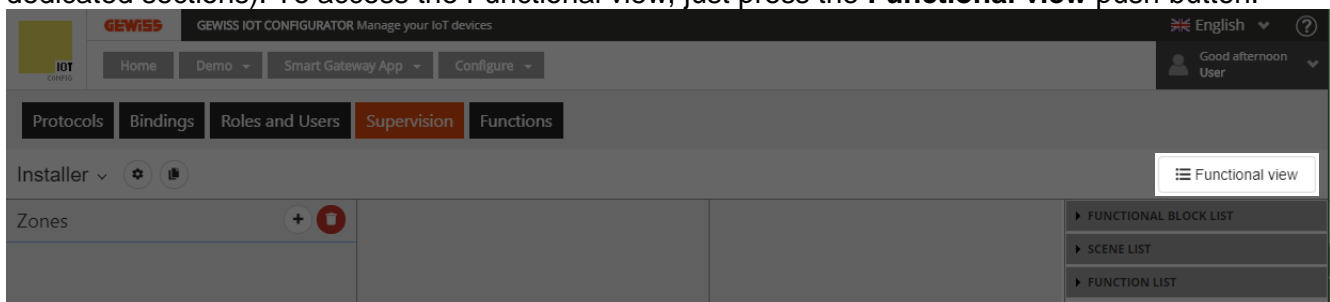
Creating the Functional view

In supervision via the app, apart from navigation based on zones/environments there are also various functional categories that the elements are grouped under.

In particular, for the functions, the modification of the configuration (timer programming, modification of the action of a scene sequence, etc.) can only be accessed in the Functional view.

By default, all the elements inserted in the Zones/Environments structure are automatically associated with the Functional view too; here, you can remove elements or add new ones that you don't want to see in Zones/Environments navigation.

In particular, for the functions that you want to make modifiable, cameras to be visualised, graphic trends and functional blocks (energy meter type) whose consumption trends you want to see, it's essential to insert them in the Functional view because they're only available in the dedicated section.
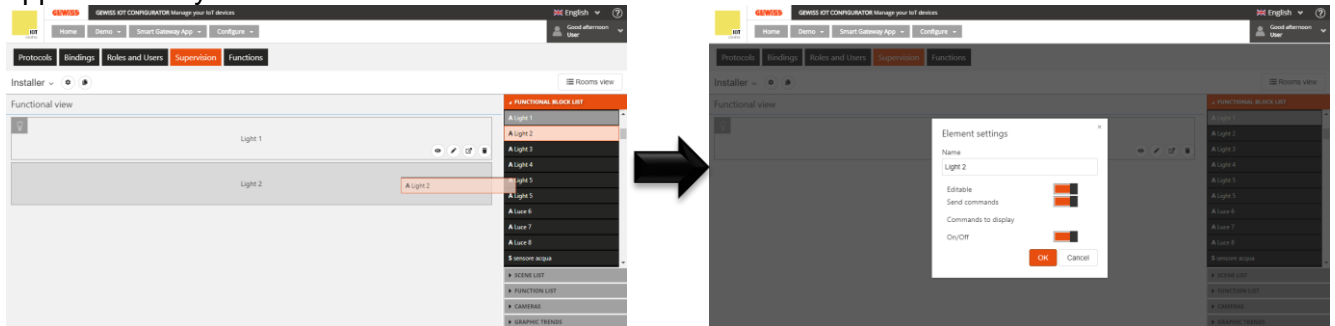
In the IoT Configurator, the Functional view is represented by a list containing all the elements that belong to it, without any distinction of the type of element (in the app on the other hand, they have dedicated sections). To access the Functional view, just press the **Functional view** push-button.

When you have selected the Functional view, the supervision design area will display all the elements that have been "enabled" for viewing in the relative functional categories.
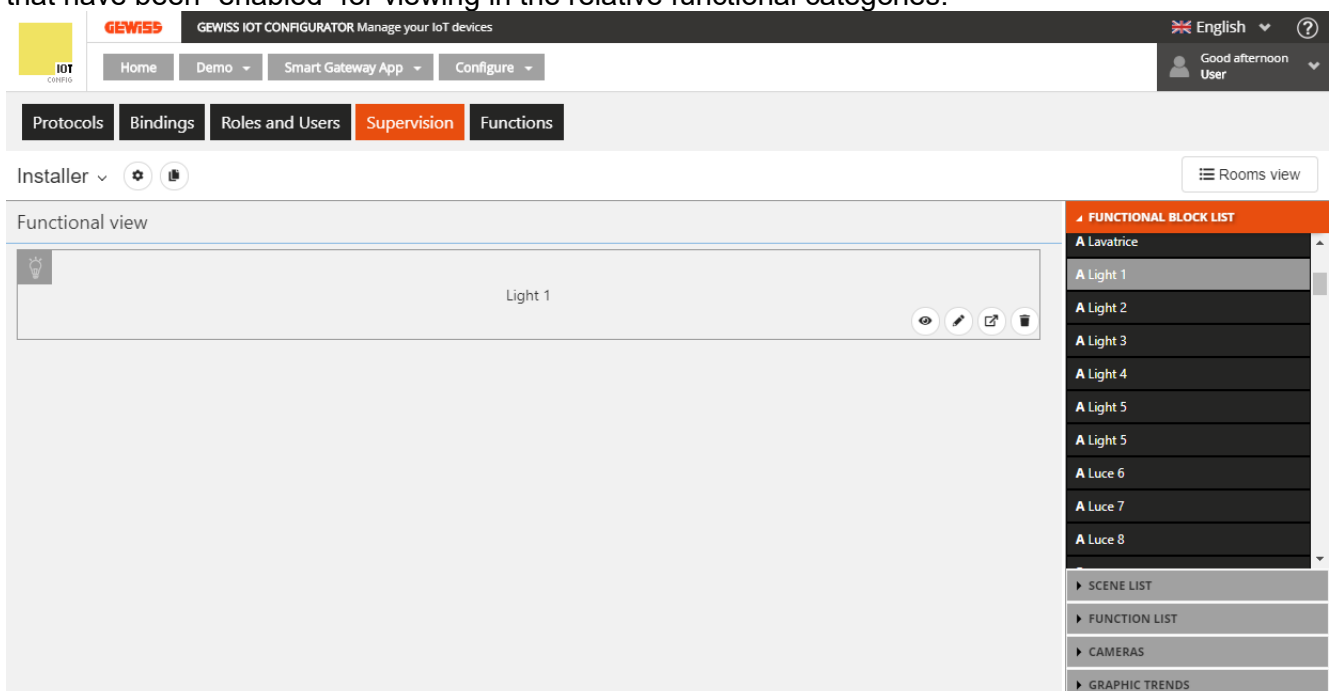
To add an element (functional block, scene, function, cameras, and graphic trends) in the Functional view, select the element from the list (right-hand column) and drag it into the supervision design area; release the object in the required visualisation position. At the end of this operation, a pop-up menu will appear where you can select:

- name of the element that will be visualised in the supervision via app (can be different from the name used in configuration)
- enabling of the modification of the element via app: if the **Modifiable** box is ticked, the element can be modified via the app; otherwise, the element will be visible but its configuration can't be altered
- enabling of the sending of element-related commands from the app: if the **Send commands** box is ticked, all the graphic widgets of the element that involve the sending of commands will be enabled to do so; otherwise, the widgets will indicate the status but it will be impossible to send commands.
- Selection of the element commands/statuses that you want to make available on the app; the list of commands available will depend on the type of element you are configuring

Once the settings have been confirmed, the element will be shown in the list of the Elements of the Functional view, in the position defined when you dragged it.
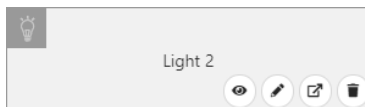
In the middle of the element you can see the name assigned to it and visualised in the supervision via app.

- **Go to the object** push-button for passing directly to the page dedicated to the element (for making quick modifications)
- **Modify** push-button for reactivating the pop-up where you can modify the name, enable the modification and sending of commands, and select the commands to be visualised
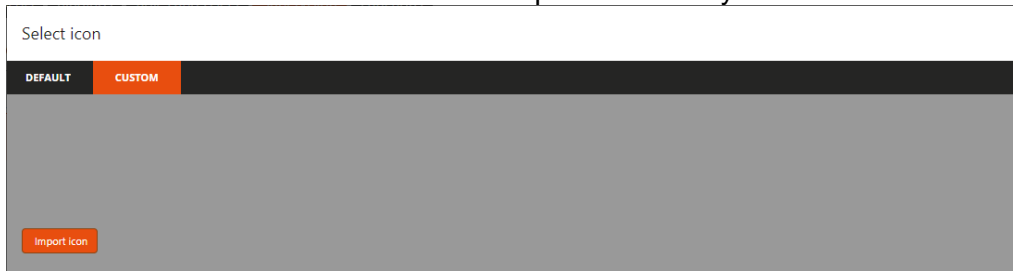- **Delete** push-button for eliminating the element from the Functional view

At the top left, there's the icon representing the element in the supervision via app. If no icon is defined, the default one will appear during configuration; when the push-button is pressed, a pop-up appears where you can select the icon.
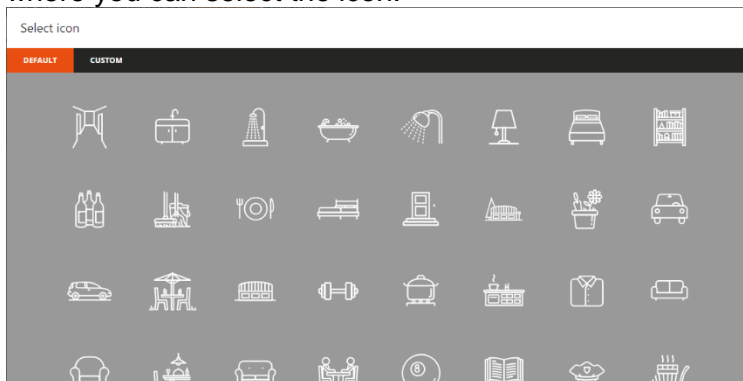
The pop-up is formed of two menus: DEFAULT and CUSTOM.

The first menu contains the icons made available by GEWISS.

The second menu shows all the icons imported in the system.

To import an icon, press the "Import icon" push-button and then select the required icon.
Maximum file size: 1 MB
File extensions supported: svg/png/jpeg

An element added to the Functional view can't be added to the Zones/Environments structure; if you want to do this, you must first of all remove it from the Functional view and then add it to an environment (after which it will be automatically reinserted in the Functional view).

The order in the list of elements of the Functional view determines the order of visualisation in the dedicated sections of the app (it can be modified directly from the app); to move an element, just select it and drag it to the required position.

UPDATES

The GEWISS IoT Configurator and the configuration section of the Smart Gateway are automatically updated as soon as a new version is released.
To see the version, press the icon on the bar at the top.

When you press the icon, a drop-down menu will appear with both versions.

SAT

+39 035 946 111
8.30 - 12.30 / 14.00 - 18.00
lunedì ÷ venerdì - monday ÷ friday

+39 035 946 260
24h

sat@gewiss.com
www.gewiss.com