GPM SCADA

2023.3 - User Guide

Table of Contents

Get started

Software requirements

GPM SCADA is a remote application and has no CPU and RAM minimum requirements. However, the remote desktop client that connects you to GPM SCADA does use your machine's resources.

The requirements to use GPM SCADA through a remote desktop client are the following:

- A remote desktop client, such as Remote Desktop Connection in Microsoft Windows.
- Connection speed: 2Mb/s in download.
- Screen resolution: 1024 x 768.

Connect to GPM SCADA

GPM SCADA is hosted on a remote server and uses Active Directory to manage users. A GPM representative will give you the Remote Desktop Connection (RDP) file and the credentials to connect to your GPM SCADA instance.

There are two methods to connect to GPM SCADA depending on your system:

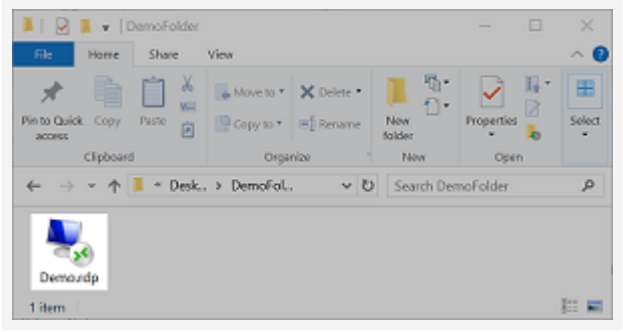
- Connect from Windows
- Connect from Mac

Connect to GPM SCADA from Windows

To connect to from a Windows device, follow these steps:

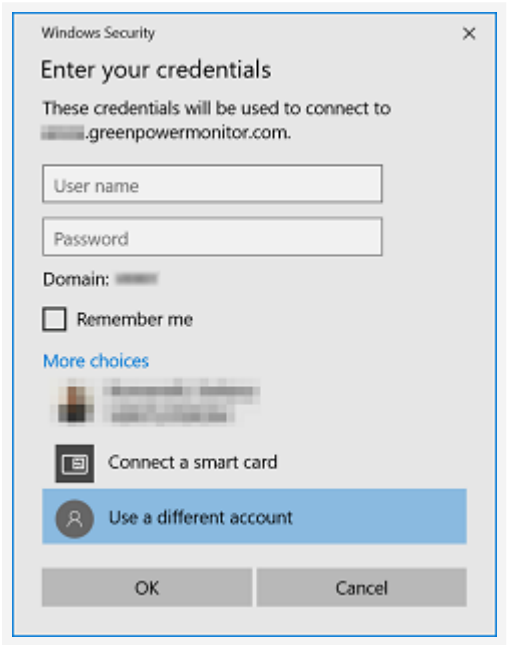
1. Open the RDP file provided by your GPM representative.

RDP file



Result: The Remote Desktop Connection window opens:

Remote Desktop Connection

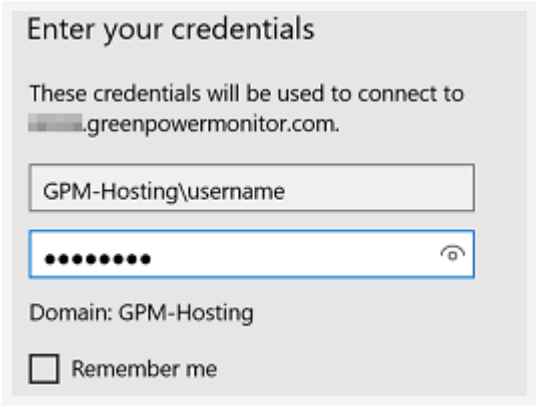


2. In the *User name* and *Password* fields, enter your Active Directory username and password.

> ⓘ NOTE: Use the format `DOMAIN\username`, where "DOMAIN" is the domain from the **Active Directory**. The domain name is saved for future log-ins, but you can change it by following the alternative to this step below.

OR: If it is the first time you are logging in, or if you want to access GPM SCADA with a different username, on the **Windows Security** dialog, click **More choices** and select **Use a different account**.

Remote log-in example



Result

You are now connected to GPM SCADA.

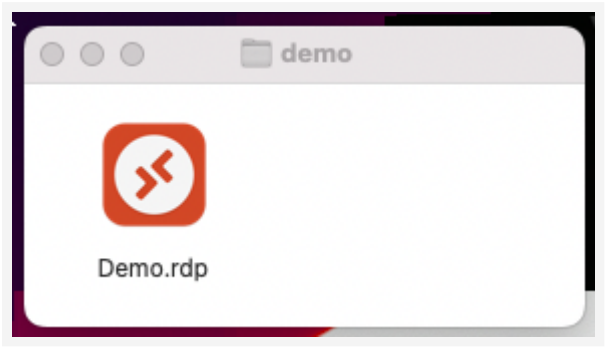
Connect to GPM SCADA from Mac

To connect to GPM SCADA from a Mac device, follow these steps:

> ⓘ NOTE: To establish a remote connection, you first need to install the Microsoft Remote Desktop app from the Mac App Store.

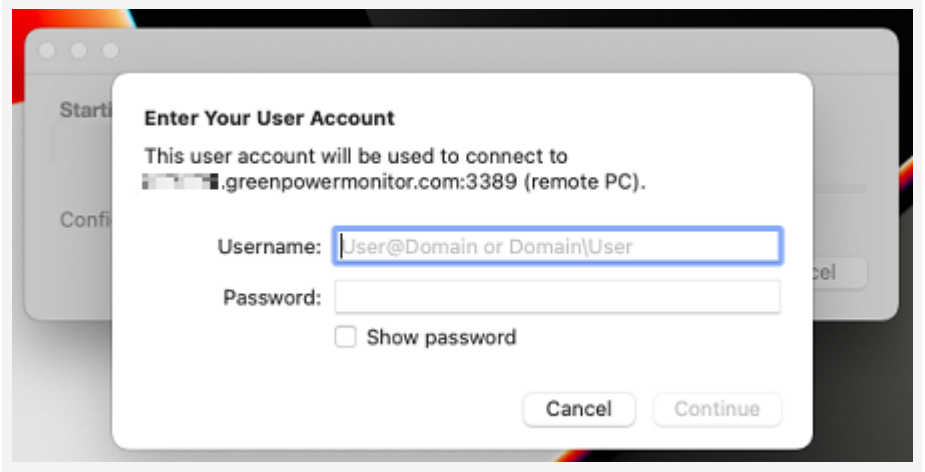
1. Open the RDP file provided by your GPM representative.

   RDP file

   Result: The Remote Desktop Connection window opens:

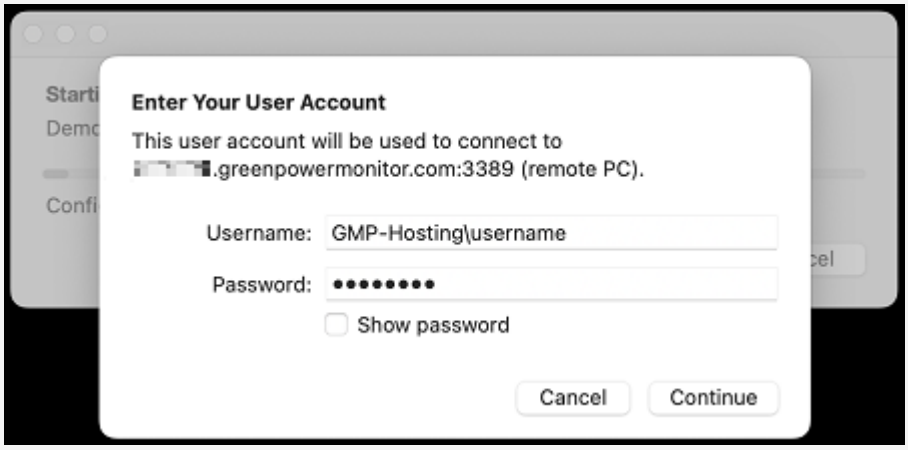
   Remote Desktop Connection

2. In the *User name* and *Password* fields, enter your Active Directory username and password.

   > ⓘ NOTE: Use the format `DOMAIN\username`, where "DOMAIN" is the domain from the **Active Directory**.

Remote log-in example



Result

You are now connected to GPM SCADA.

Disconnect from GPM SCADA

You can disconnect from GPM SCADA by signing out of your remote desktop connection.

There are two methods to disconnect from GPM SCADA depending on your system:

- Disconnect from Windows
- Disconnect from Mac

Disconnect from GPM SCADA from Windows

To disconnect from GPM SCADA from a Windows device, follow these steps:

1. With the Remote Desktop Connection window active, press **Ctrl + Alt + End**.

   Result: The remote connection options open:

   Remote connection options

2. Click **Sign out**.

Result

You are now disconnected from GPM SCADA.

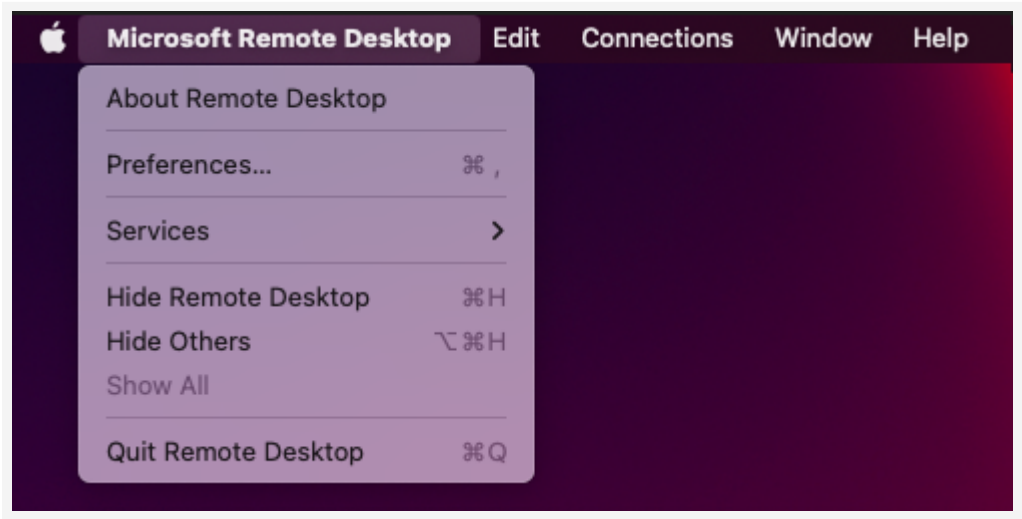
Disconnect from GPM SCADA from Mac

To disconnect from GPM SCADA from a Mac device, follow these steps:

1. With the Remote Desktop Connection window active, select **Microsoft Remote Desktop** and then **Quit Remote Desktop**.

   Result: The remote connection options open:

   Remote connection options

2. Click **Sign out**.

Result

You are now disconnected from GPM SCADA.

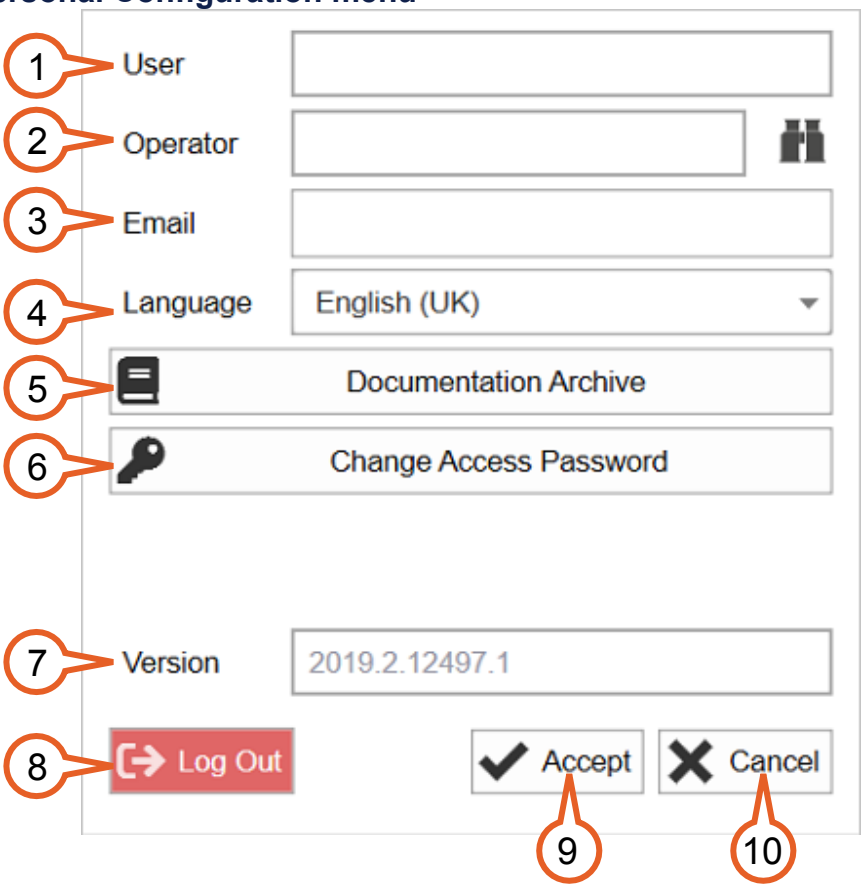
Personal Configuration

The Personal Configuration menu allows you to set the application language, your contact email, access information, and other data to customize your user experience. Additionally, you can see the application version and download the latest user documentation.

> ⓘ NOTE: When you first access the Personal Setup section after a new version update, a pop-up appears that prompts you to view the release notes for the version. If you decide to view them later, you can access them through the Documentation Archive.

Click the icon on the **Upper Bar** to access the Personal Configuration.

Personal Configuration menu



1. User: displays your username. This field cannot be changed.
2. *Operator*: click to link your Active Directory user to an operator profile. Operator users have a set of characteristics, such as user status and phone number, that are used to track alarm and ticketing operations.

   Click the icon to change the operator

3. Email: displays the e-mail account linked to the application.
4. **Language**: click the drop-down menu to select the language for the user interface.
5. **Documentation Archive**: click to access the latest documentation. In the Documentation Archive window, double-click a document to open it in the PDF viewer (for example, the release notes for the latest update of GPM SCADA).

   When you first access the **Personal Setup** after a new version release, a pop-up appears to give you the option to view the latest release notes. If you choose to view them later, they are available in the Documentation Archive.
6. **Change Access Password**: select this option if you want to change your password. In the following dialog you are prompted to enter your current password, your new password, and confirm the new password.

   > ⓘ NOTE: For security reasons, your system administrator may enforce a password expiration policy. The application notifies you when the expiration date is near.
7. Version: displays the current version number of the application.
8. **Log Out**: click to close the Remote Desktop Client. This action leaves your user session open and you will already be logged in next time you open the Remote Desktop Client.
9. **Accept**: click to save your changes.
10. **Cancel**: click to close the window without saving your changes.

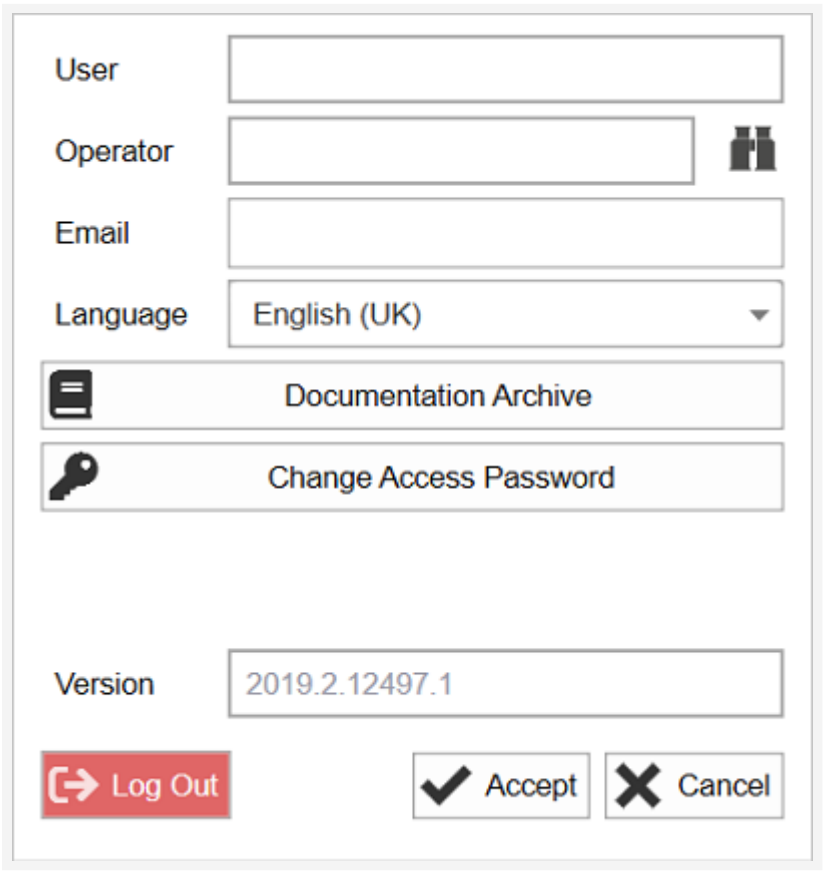
Link user accounts to Operator profiles

To link your user account to an operator profile, follow these steps:

1. Click the icon on the top navigation bar to open your personal configuration.

   Result: The Personal configuration dialog appears:

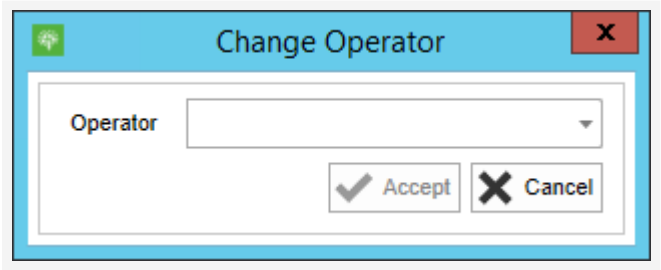
   Personal configuration menu



2. Click the icon next to the *Operator* field.

   Result: The **Change operator** dialog appears:

   Change operator dialog



3. Click the **Operator** drop-down menu and select your name from the list, then click ✓ **Accept**.
4. In the **Personal configuration** dialog, click ✓ **Accept**.

Result

The operator profile is linked to your Active Directory account.

Quick tour

GPM SCADA is a modular application. The different modules allow you to perform various browsing and operational tasks.

To see a list of all the modules available in GPM SCADA, see the Modules section.

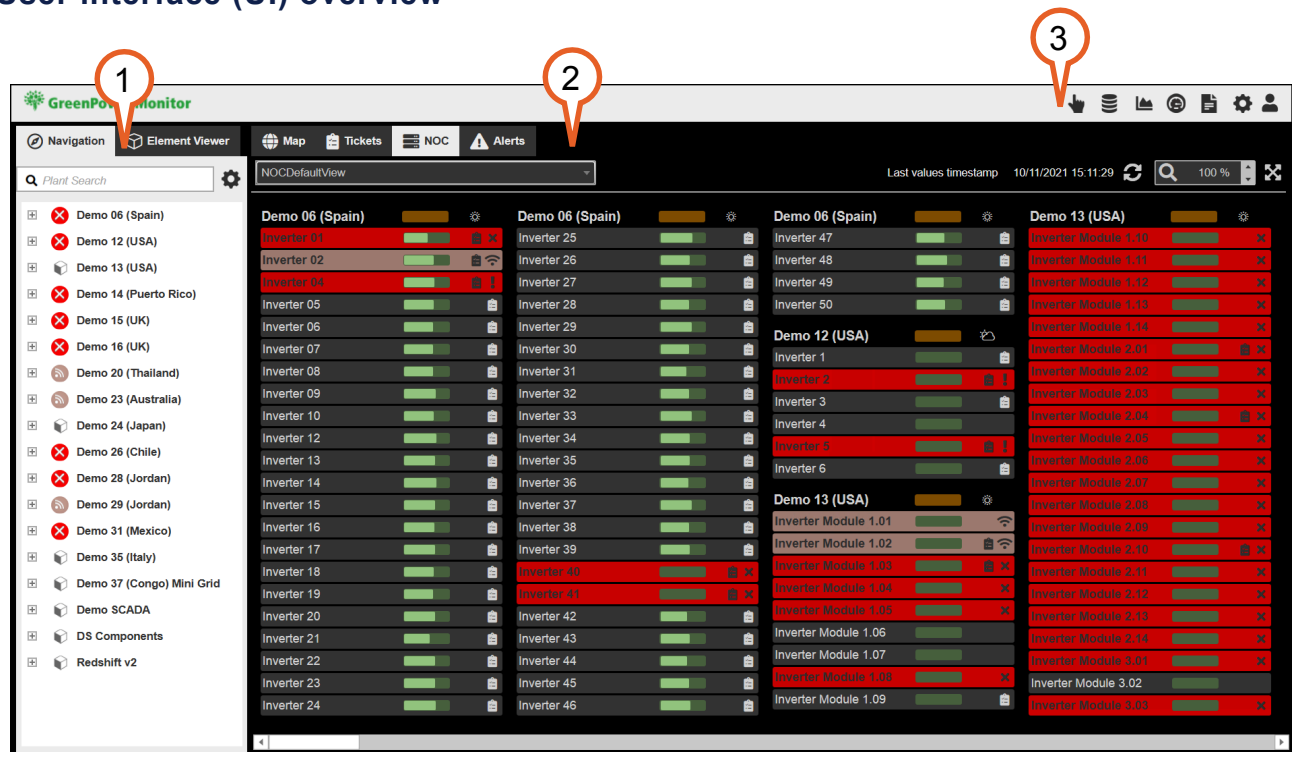
You can access these modules from three main areas of the user interface:

1. Side panel
2. Content area
3. Upper bar

In some areas, tabs are used to condense multiple modules within the same interface.

> ⓘ NOTE: Access to each module is configured based on the intended user needs. Some modules described in this document may not be available to you if your setup does not require them.

User interface (UI) overview

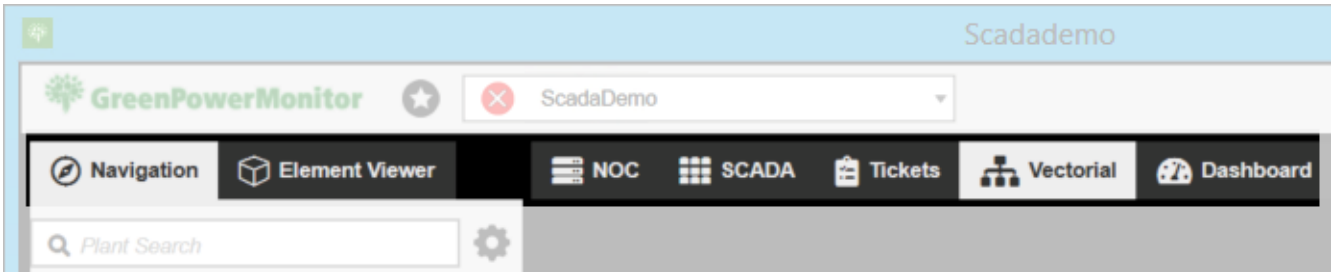


1. Side panel: contains the Navigation and the Element Viewer modules, displayed as tabs. For more information, see Navigation Module and Element Viewer Module.
2. Content area: displays content from the main modules: NOC, Map, Tickets, Alarms, and Plant Dashboard.
3. Upper bar: contains buttons that provide access to additional modules, the application setup, and the personal configurations for the user. For more information, see Modules and User Configuration.

Tab navigation

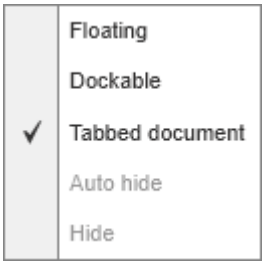
The user interface of GPM SCADA contains top-level navigation tabs to facilitate your navigation through various operational modules.

Tabs in the UI



Each tab has a context menu that allows you to customize its position and appearance in the application. You can access the context menu by right-clicking a tab.

Tab-level context menu



- **Floating**: move the tab outside of the main window and:
  - Drag it to a different area of the screen.
  - Drag it to another screen.
  - Set the tab to full screen by double-clicking the tab's heading.
- **Dockable**: Move the tab outside of the main window and:
  - Drag it to a different area of the screen.
  - Drag it to another screen.
  - Drag it to one of the pop-up icons to anchor it to a specific area of the screen.
  - Set the tab to full screen by double-clicking the tab's heading.
- **Tabbed document**: When a tab is set to **Dockable** or **Floating**, select this option to anchor it as the last tab in its default location.

- **Hide**: When a tab is set to **Dockable** or **Floating**, select this option to anchor the tab to its default position.
- **Autohide**: When a tab is anchored to the top or bottom of the screen, select this option to automatically hide the tab when you are not using it.

Side panel

The Side Panel contains the **Navigation** and **Element Viewer** modules, divided in tabs.

The Navigation module allows you to browse your portfolio. For more information, see the Navigation module section.

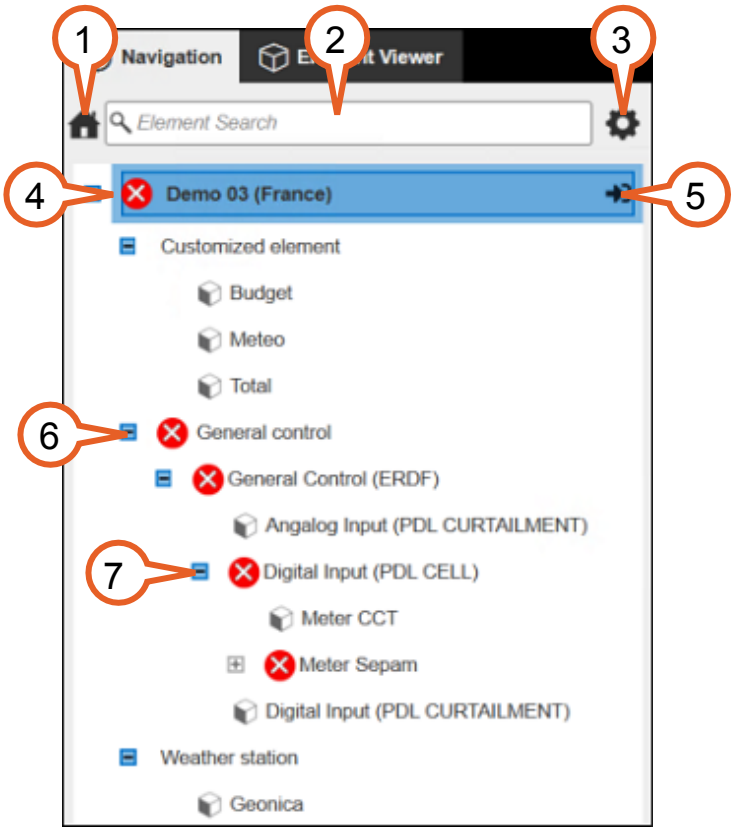
The Element Viewer module allows you to access information about elements. For more information, see the Element viewer module section.

Navigation module

The Navigation module lists the plants in your portfolio and their child elements. Each plant is organized as a cascading object to allow you to navigate the plant structure.

You can perform several tasks in your plants directly from the Navigation module. For more information, see the list of available tasks.

Navigation module



1. **Navigation home**: click to return to the portfolio list. This button is only displayed when you are navigating inside a plant.
2. *Search bar*: enter text to search for:
   - plants when you are navigating at the level of the portfolio.
   - elements when you are navigating inside a plant.
3. **Navigation settings**: click to customize the Navigation module and set the plant grouping criterion from available metadata such as country or technology.

Additionally, you can:

- Set the Plant Icon as the **Main Alarm** of the plant, or as **% Available Peak Power**
- Sort plants by **Name**, **Peak Power**, or **% Available Peak Power**.
- Arrange elements by **Production Hierarchy**, **Communication Hierarchy**, or **Device Type**.

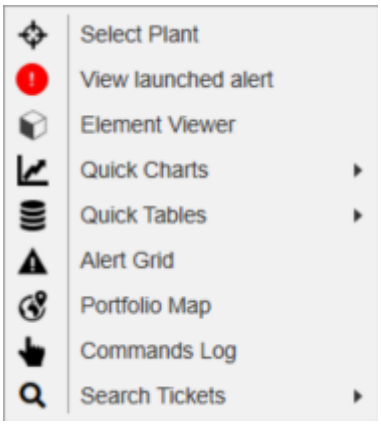
4. **Plant name and icon**: place your cursor on the icon to display information about the plant's main alarm or the percentage of available peak power.
   Click the ⚙ icon to customize the icon information. When the plant main alarm is displayed, you can click the icon to open the Alarm Information window.
5. **Open plant**: click to open the Plant Module in the Content Area.
6. **Expand/collapse**: click to expand or collapse plant details.
7. **Element icon**: click to display the element information on the Element Viewer. Right-click an element to to open the context menu for it and see the available options.

   When there is an active alarm for the element, an alarm icon is displayed. You can click the alarm icon to open the Alarm Information window and place your cursor on the icon to see the alarm message. When the icon is set to display the **% Available Peak Power**, placing your cursor on the icon displays the percentage information.
8. Element name: displays the name of the element.

Context menu

The context menu allows you to perform quick actions from the Navigation module. You can activate it by right-clicking a plant or an element in the module.

Context menu: plant

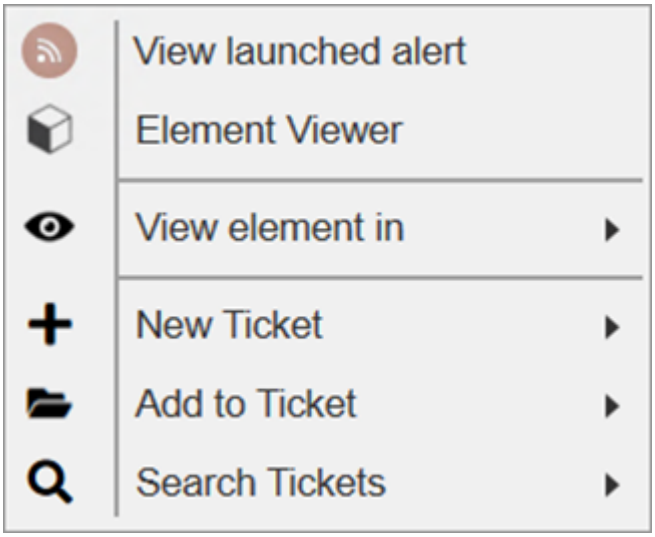


- **Select plant**: Open the Plant module for the selected plant.
- View launched alarm: Open the related alarm in the Alarm information window.
- **Element viewer**: Open the Element Viewer for the selected plant.
- **Quick charts**: Open the selected custom query in the Linear Chart Viewer module.

  Since the back-end processes charts differently, the **Quick Chart** menu contains two sub-menus:

  - **Custom Charts**: contains charts that are complex to build because they are built specifically for your portfolio and can query any parameter.
  - **Default Charts**: contains charts that query only generic parameters that every portfolio has, such as energy, PR, or availability.
- **Quick tables**: Open the selected custom query in the Data Viewer module.
- **Alarms table**: Display all the related alarms in the Alarms module.
- **Portfolio map**: Display the plant on the Map module
- **Commands log**: Display related commands in the Commands Control module.
- **Search tickets**: Choose a ticket type to open the related tickets in the Tickets module.

Context menu: element



- **View launched alarm**: open the related alarm in the Alarm information window.
- **Element viewer**: open the Element Viewer for the selected element.
- **View element in**: display all the related alarms in the Alarms module or the Commands Control module.
- **New ticket**: select a new ticket type to create a new ticket and automatically pair the selected element to it. For more information, see Create maintenance tickets from elements, Create task tickets from elements and Create data correction tickets from elements.
- **Add to ticket**: Place your cursor here, enter a ticket ID in the *Search* field, and press **Enter** to add teh selected element to an existing ticket.
- **Search tickets**: Choose a ticket type to open the related tickets in the Tickets module.

Working with the Navigation module

The Navigation module allows you to perform several tasks with different features and tools.

Tickets

- Create maintenance tickets.
- Create task tickets.
- Create data correction tickets.
- Add elements to existing tickets.

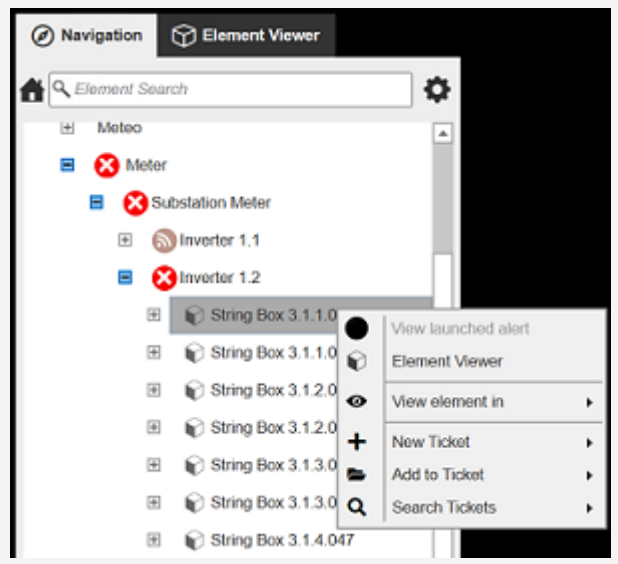
Create new maintenance tickets from the Navigation module

To create a new maintenance ticket from the Navigation module, follow these steps:

1. In the Navigation module, navigate to the level of the element for which you want to create the ticket, then click the icon of the element to select it. Then, right-click the selected element to open the Context menu:

   Result: The Context menu appears:

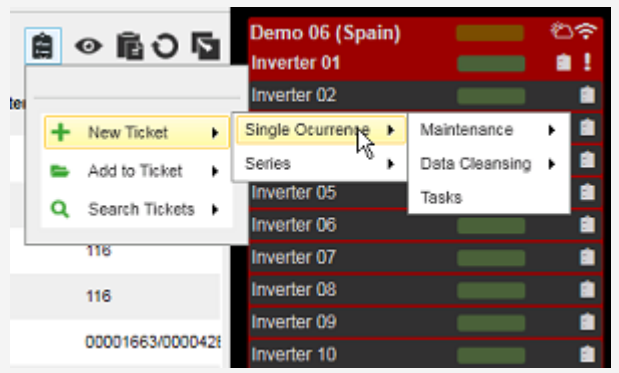
Context menu for an element in the Navigation module



2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.

   Result: The ticket type panel opens:

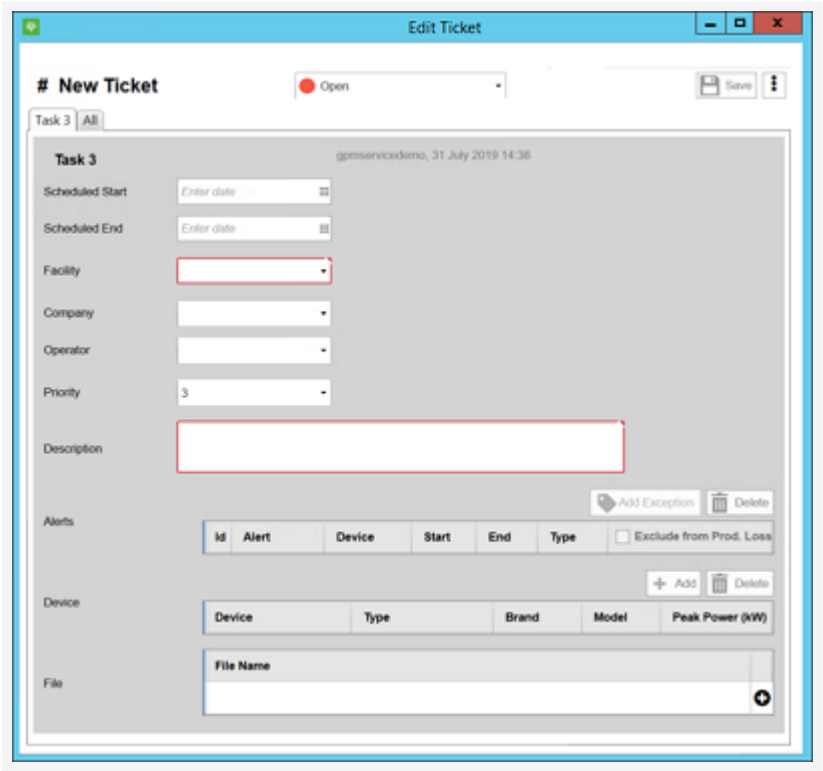
Ticket types



3 In the menu, select **Maintenance**, then select the **GPM Default** template.

Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog



4 In the **Edit Ticket** dialog, enter the information for the ticket:

a **Scheduled Start**: select the date on which the ticket starts.

b **Scheduled End**: select the date on which you expect the ticket to close.

c **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.

d **Company**: select the company that must resolve the issue related to the ticket.

e **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

h **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

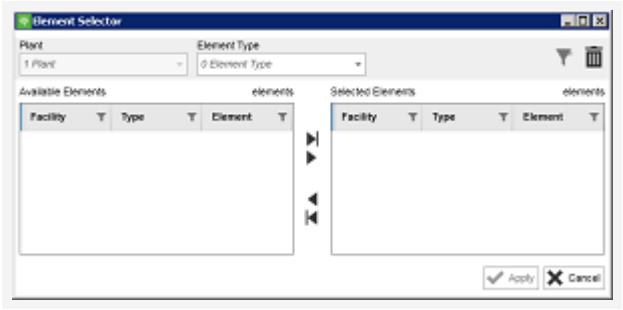
i (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information,

see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

5. (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a. Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b. In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c. Click **Apply**.
Result: The element is added to the list.

6. (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).
7. Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

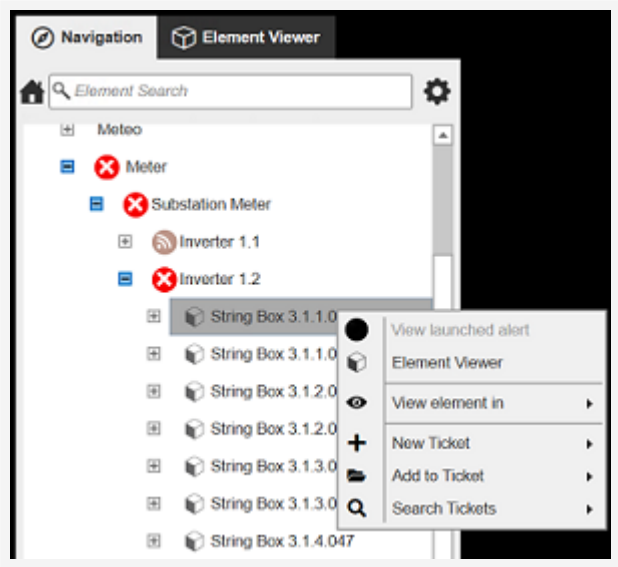
Create task tickets from the Navigation module

To create a task ticket from the Navigation module, follow these steps:

1. In the Navigation module, navigate to the level of the element for which you want to create the ticket, then click the icon of the element to select it. Then, right-click the selected element to open the Context menu:

   Result: The Context menu appears:

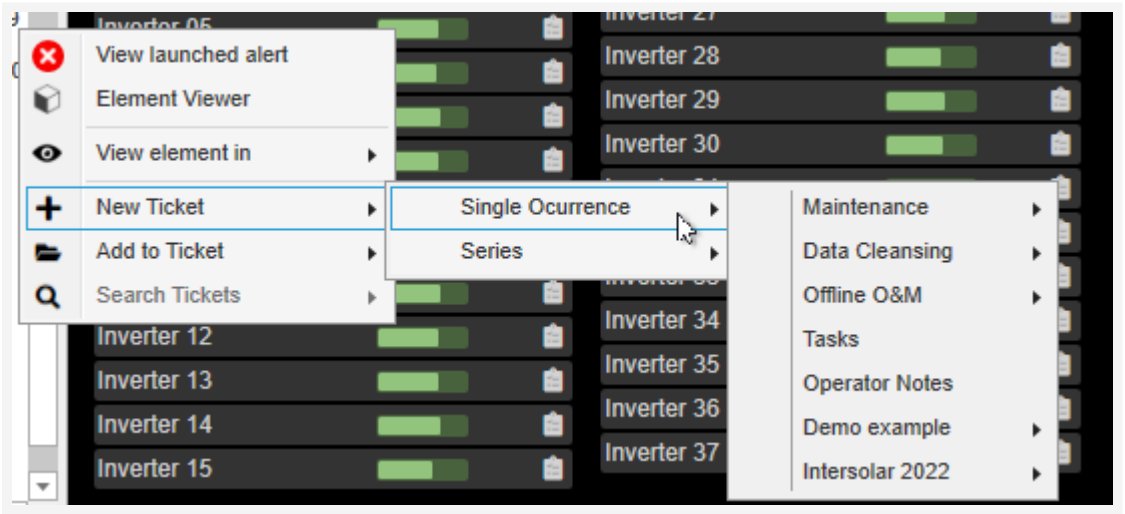
   Context menu for an element in the Navigation module

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.

Result: The ticket type panel opens:

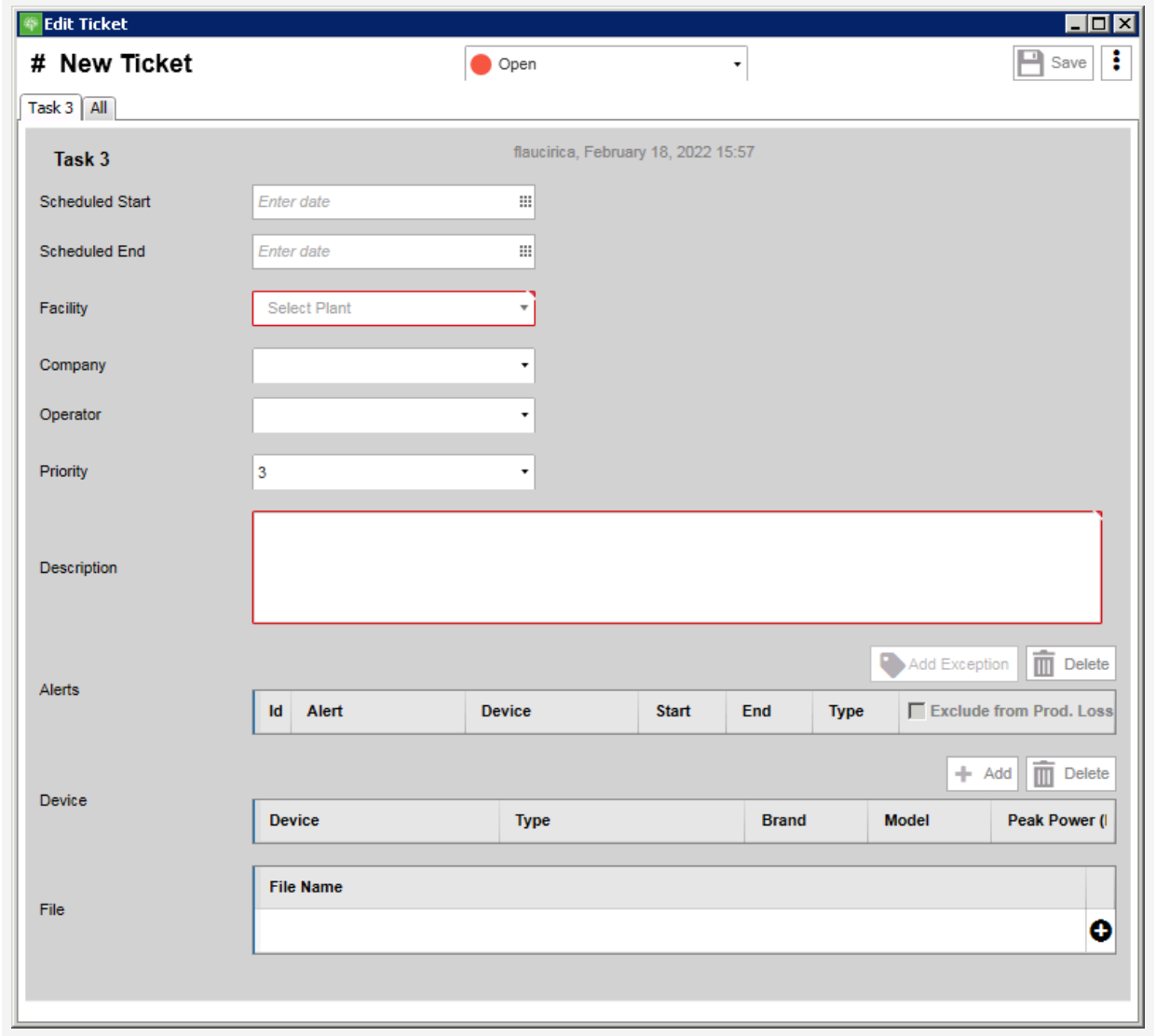
Ticket types



3. In the menu, select **Task**, then select the **GPM Default** template.

Result: The **Edit** ticket dialog appears:

Edit ticket dialog (task)



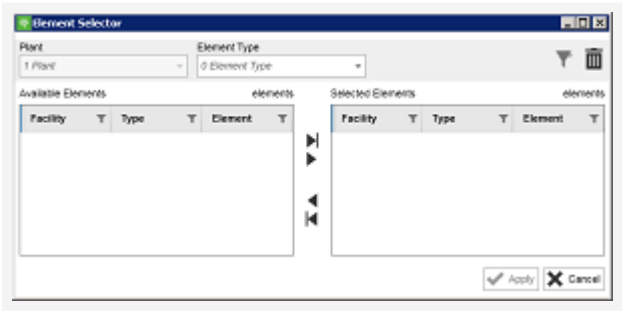
4. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

5 (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c Click **Apply**.
Result: The element is added to the list.

6 (Optional) In the File section, click the ⊕ icon to open the Import Box and import a file.

Import box

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

Result: The file is imported to the ticket.

7. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition

b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

8. Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

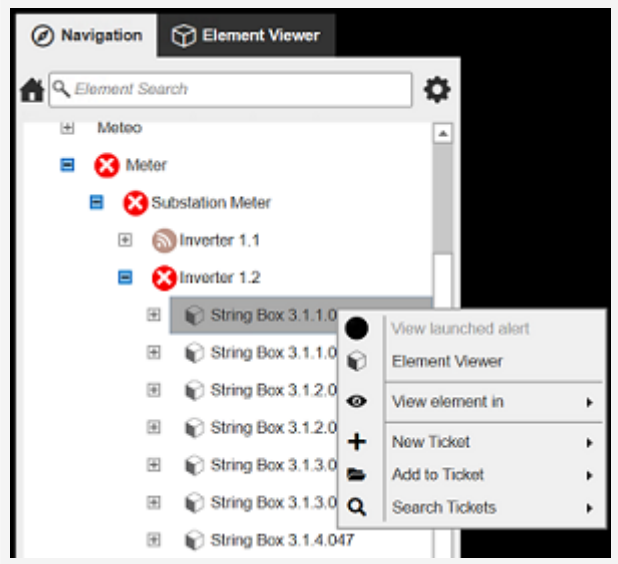
Create data correction tickets from the Navigation module

To create a data correction ticket from the Navigation module, follow these steps:

1. In the Navigation module, navigate to the level of the element for which you want to create the ticket, then click the icon of the element to select it. Then, right-click the selected element to open the Context menu:

   Result: The Context menu appears:

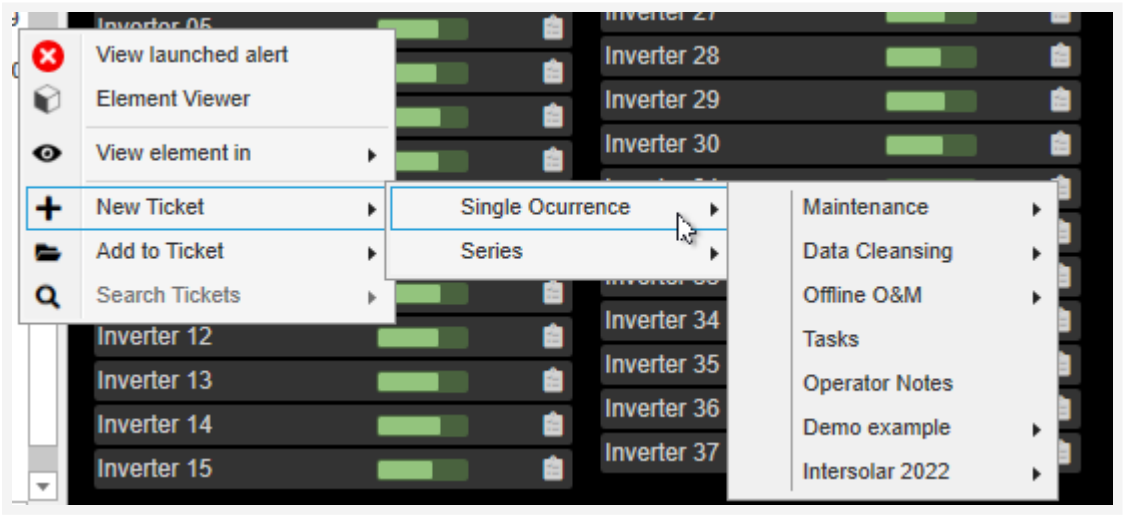
   Context menu for an element in the Navigation module

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.

Result: The ticket type panel opens:

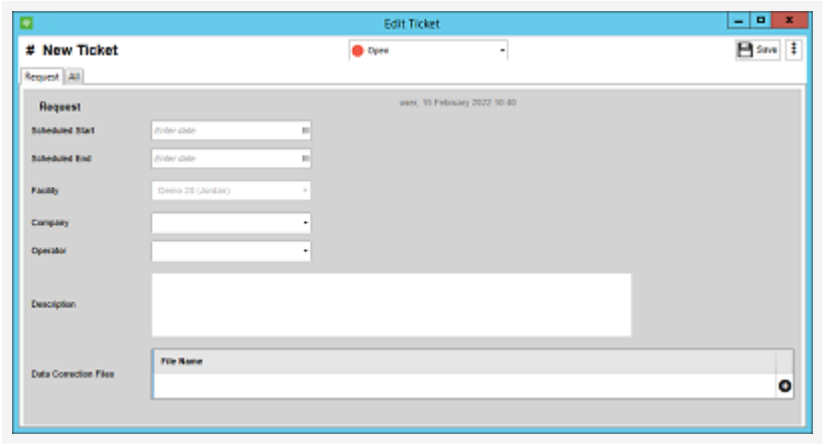
Ticket types



3. In the menu, select **Data Cleansing**, then select **GPM Data Correction**.

Result: The Edit ticket dialog appears:

Edit ticket dialog (Data cleansing)

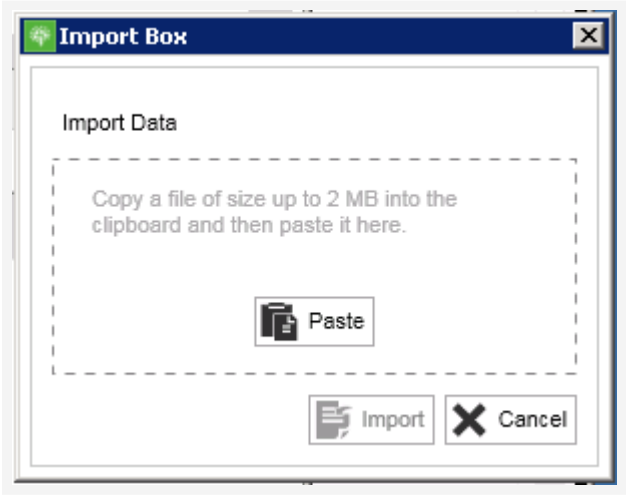


4. In the Request section, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f *Description*: enter a description in the text input field.

5 In the Data Correction Files section, click the ⊕ icon to open the Import Box and import an XLS file from which to add the corrected data:

Import Box



ⓘ NOTE: You can only import XLS files.

a In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b In the Import Box dialog, click **Paste**.

c Click **Import**.

Result: The file is imported to the ticket.

6 (Optional) To add a note, follow these steps:

a Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition

b Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

7 Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

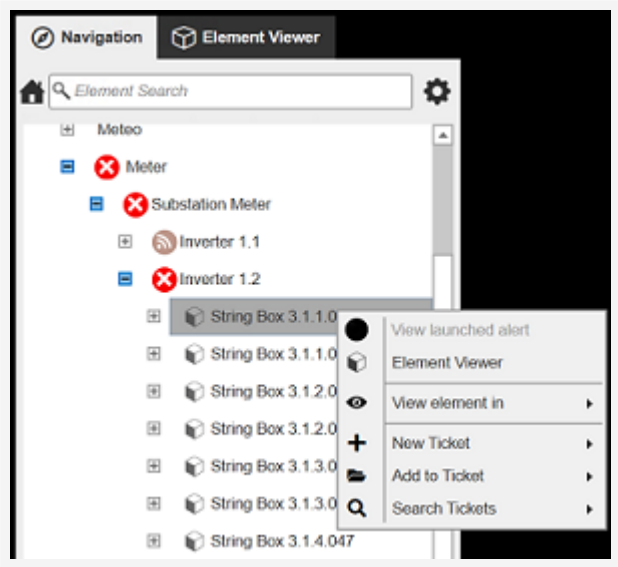
Add elements to existing tickets from Navigation

To add an element to an existing ticket from the Navigation module, follow these steps:

1. In the Navigation module, navigate to the level of the element for which you want to create the ticket, then click the icon of the element to select it. Then, right-click the selected element to open the Context menu:

   Result: The Context menu appears:

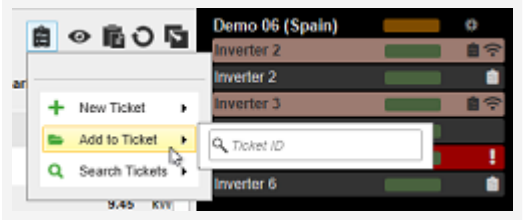
Context menu for an element in the Navigation module



2. Hover over **Add to Ticket** and enter the *Ticket ID* of an existing ticket (for example, "1543549623"), and press **Enter**.

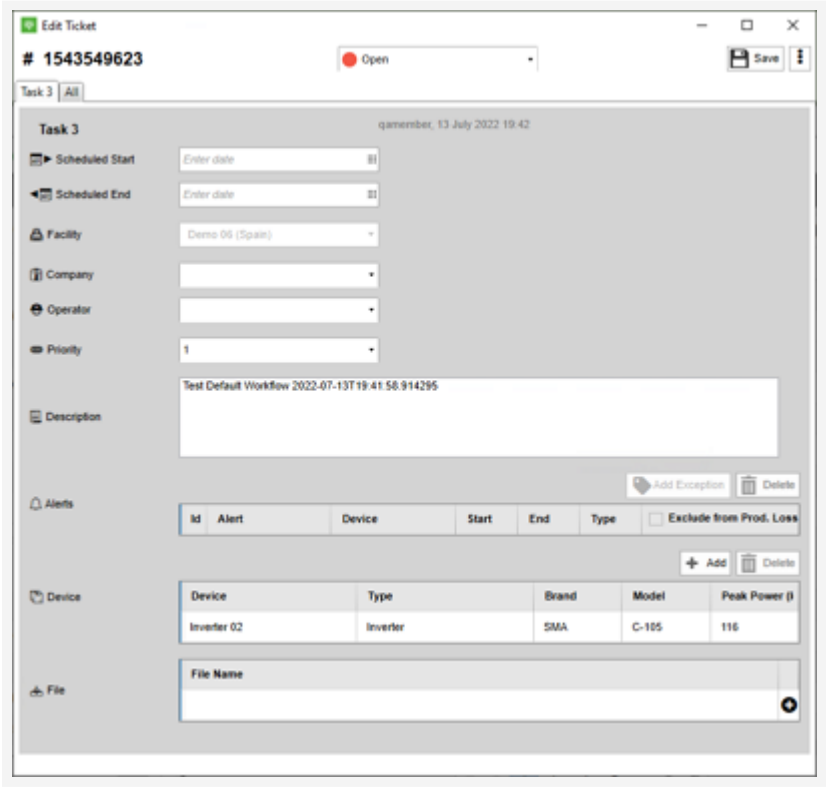
> REMEMBER: You must enter the ID of an existing ticket. If no ticket exists, follow the instructions to create a new ticket from the Element viewer.

Ticket menu



Result: The **Edit ticket** dialog appears, displaying the selected element in the Element field:

Edit ticket dialog



3. (Optional) Edit any other fields of the ticket you want to change.
4. Click **Save**.

Result

The element is added to the ticket and any other changes are saved.

Element viewer module

The Element Viewer module is a dynamic tab that allows you to quickly monitor, analyze, and perform maintenance on a selected plant or device.

> ⓘ NOTE: The information displayed on the Element Viewer refreshes automatically when GPM SCADA receives data from the selected element.

You can perform tasks to manage elements directly from this module.

Element Viewer module

1. Heading: When you select a:
   - **Plant**: displays the plant name, coordinates and peak power.
   - **Device**: displays the device name, the plant it belongs to, and its peak power. If there is an active alarm on the selected element, you can click the alarm to open the Alarm Information window.
2. **Tabs**: select a tab to display its content on the Main Area. For further information,

see the following sections:

- Monitor
- Live values
- Parameters
- Info
- Commands

3. **Quick actions**: take quick actions related to the selected plant or device.For further information, see Quick Actions below.
   - Ticket menu
   - View Element menu
   - Copy to Clipboard
   - Refresh
   - Copy Window
4. Main area: displays information about a selected element.

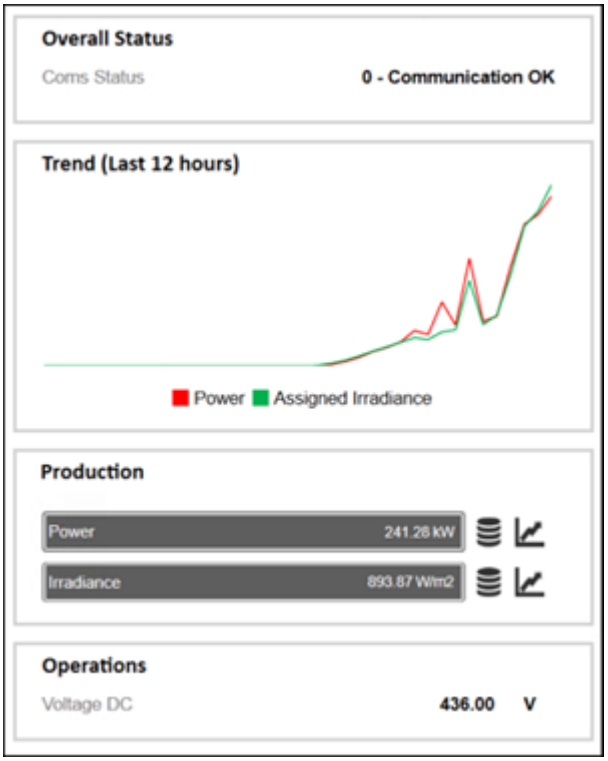
Monitor tab

The Monitor tab provides you with tools to monitor the main key performance indicators (KPIs) of a selected element. The Monitor tab tools are organized in cards and change based on the selected element type.

Access the Monitor tab by clicking the icon on the Element Viewer.

You can place your cursor on any chart heading and click the icon to display the chart KPIs in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.

Monitor tab



Live Values tab

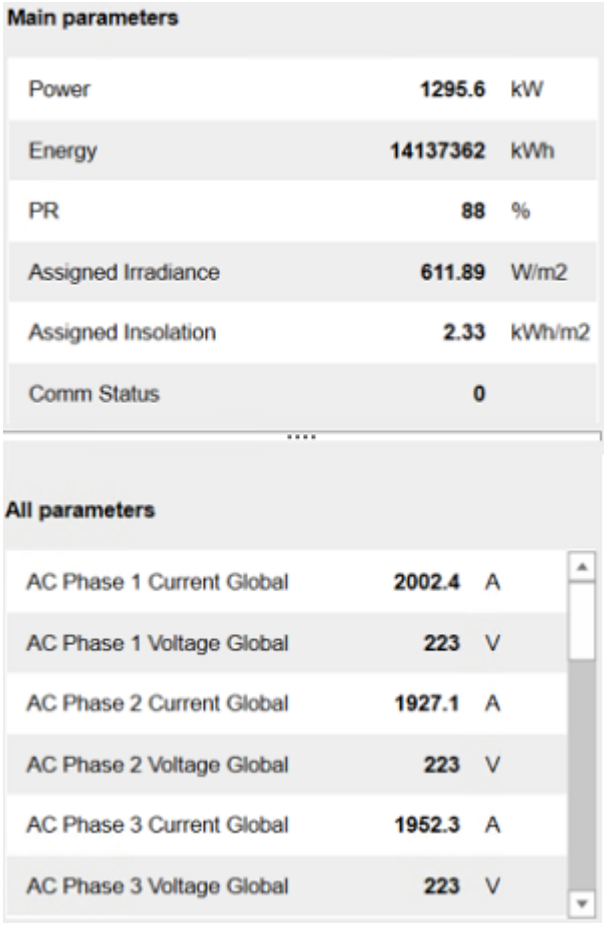
The Live Values tab allows you to monitor data that is retrieved in real time from a selected element.

> ⓘ NOTE: Real-time data availability depends on the system capability of communicating data in real time.

Access the Live Values tab by clicking the icon on the Element Viewer.

You can place your cursor on any parameter with real-time data and click the icon to display its values in the Live Viewer module. For further information, see Live Viewer Module.

Live Values tab



> ⓘ NOTE: You can resize the **Main parameters** and **All parameters** sections by clicking and dragging the dotted line that separates them. The system saves your preferences after the change.

Section	Description
Main parameters	Most relevant parameters that are configured for the selected element. ⓘ NOTE: If you want to modify the parameters displayed here, contact your GPM representative.
All parameters	All the parameters that are retrieved from the selected element.

Parameters tab

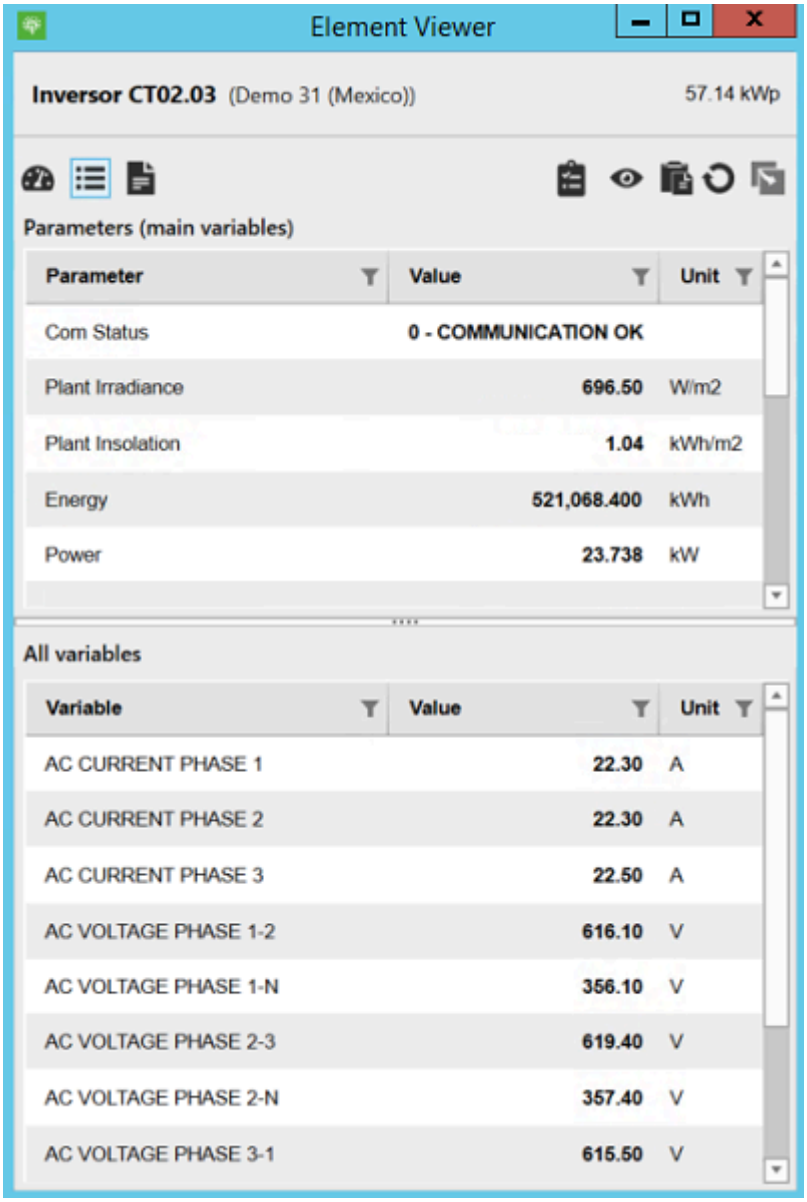
The Parameters (main variables) tab allows you to monitor data that is retrieved at regular time intervals from the selected element.

Access the Parameters tab by clicking the icon on the Element Viewer.

You can place your cursor on any parameter and click the icon to display the parameter values in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.

You can click the icon on the column headers to open the filtering options for the data on display. This allows you to find, visualize, and select specific variables. For more information about filtering options, see Advanced Filtering.

Parameters tab



ⓘ NOTE: You can resize the **Parameters (main variables)** and **Variable** sections by clicking and dragging the dotted line that separates them. The system saves your preferences after the change.

Section	Description
Main Parameters	Most relevant parameters that are configured for the selected element.

ⓘ NOTE: If you want to modify the parameters displayed here, contact your GPM representative.

All Variables	All the variables that are retrieved from the selected element. ⓘ NOTE: This section is only available for elements.

Info tab

The Info tab provides contextual information about the selected plant or device. The information is retrieved from the entity's metadata, which is customizable and can change depending on your setup.

Access the Info tab by clicking the element icon in the Navigation module or by clicking the  icon on the Element Viewer.

You can right-click a parameter and select **See History** to open the Entity Log, where you can track and manage changes made the values. For further information, see Entity Log.

Info tab

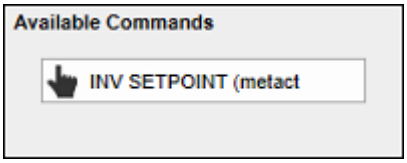
Description Parameters	
Name	INV 15B
Typology	Inverter
Description	00000DCE/000021FC/2

Commands tab

The Commands tab allows you to send pre-configured commands to one or more devices. When you click a command on the list, the **Command Execution** dialog opens.

Access the Commands tab by clicking the icon on the Element Viewer.

Commands tab



Quick Actions

The quick actions buttons allow you to easily perform basic tasks directly from the Element Viewer. Click any icon to access the available options:

- **Ticket menu**: pair the selected element to a ticket or find existing tickets related to the element.

Option	Description
New Ticket	Create a new ticket and automatically pair the selected element to it: ▪ Create maintenance tickets ▪ Create task tickets ▪ Create data correction tickets
Add to Ticket	Choose a ticket type to open the related tickets in the Tickets module.
Search Tickets	Choose a ticket type to open the related tickets in the Tickets module.

- **View Element menu**: display information related to the selected element.

> ⓘ NOTE: The options available when you click the View Element button depend on the selected element.

Option	Description
Alarm Grid	Display related alarms on the Alarms tab of the Content Area.
Search Tickets	Choose a ticket type to open the related tickets in the Tickets module. ⓘ NOTE: This option is only available when you are viewing a plant in the Element Viewer.
Portfolio Map	Display the plant on the Portfolio Map tab of the Content Area.

ⓘ NOTE: This option is only available when you are viewing a plant in the Element Viewer.

Commands Log	Display related commands in the Commands Control module.

- **Copy to clipboard**: click to copy all the parameters and values on display to your clipboard.
- **Refresh**: click to refresh te information displayed in the tab.
- **Copy Window**: click to open a copy of the Element Viewer module in a separate window.

Element management

You can manage elements directly from the Element Viewer.

Tickets

- Create maintenance tickets
- Create task tickets
- Create data correction tickets
- Add elements to existing tickets

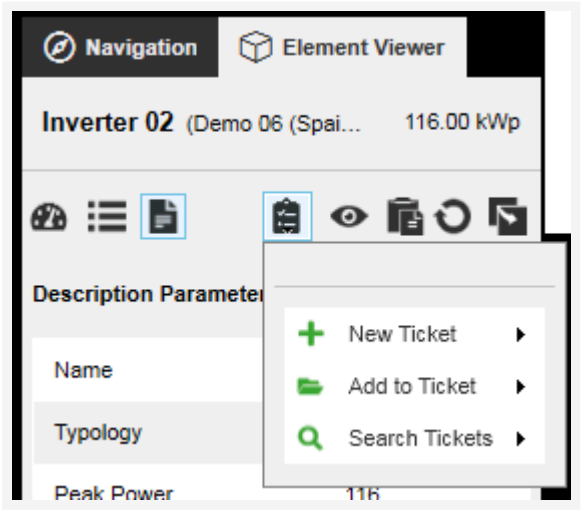
Create maintenance tickets from elements

To create a maintenance ticket from an element, follow these steps:

> ⓘ NOTE: Information contained in Maintenance tickets is customizable. Each customization is stored as a template. The following instructions use the **GPM Default** template.

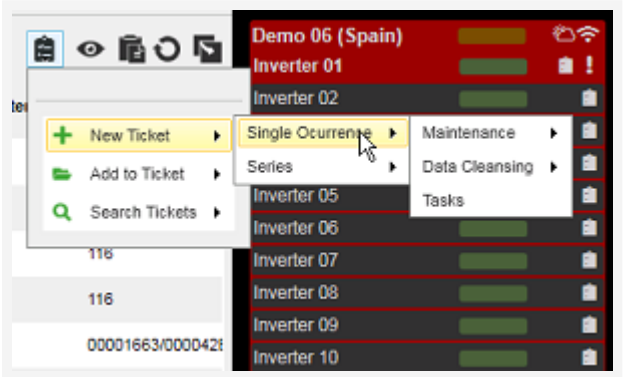
1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

   ### Ticket menu

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.
   Result: The ticket type panel opens:

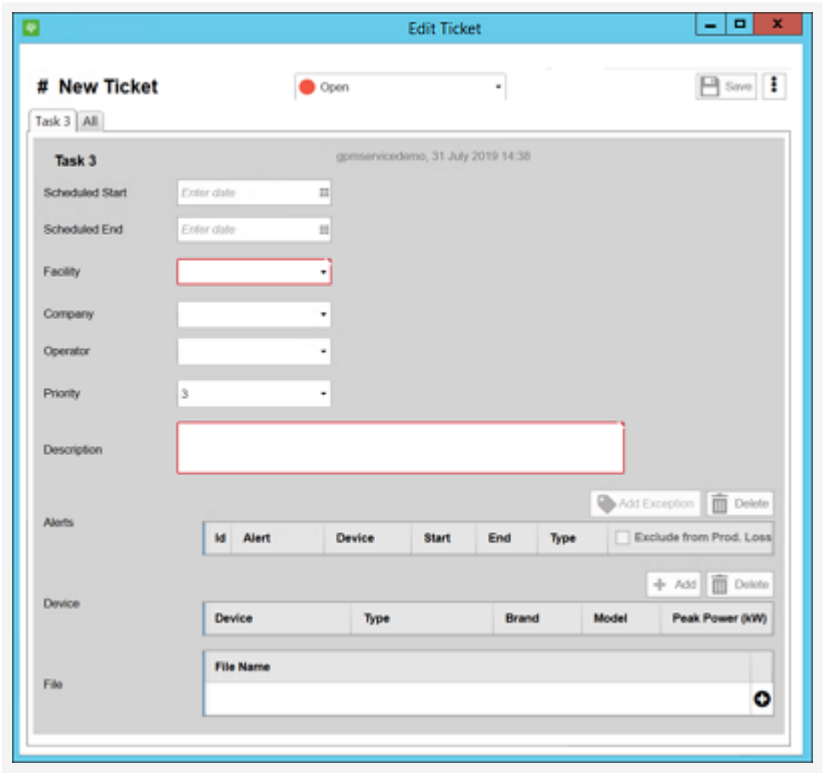
   ### Ticket types

3. In the menu, select **Maintenance**, then select the **GPM Default** template.

Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog

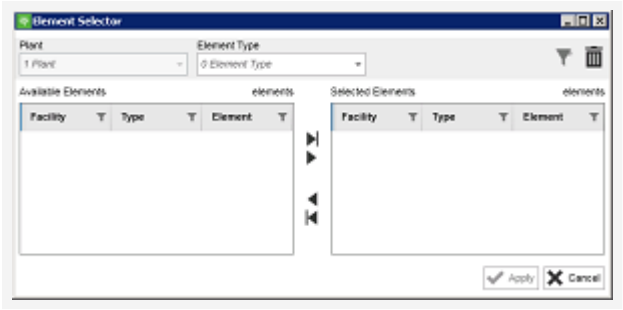


4. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. **Priority**: select a priority from the drop-down list.
      Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.
   g. *Description*: enter a description in the text input field.
   h. **Device**: if you have selected a plant, you can specify the elements to which the ticket applies
   i. (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

5. (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a. Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b. In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c. Click **Apply**.
Result: The element is added to the list.

6. (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).
7. Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

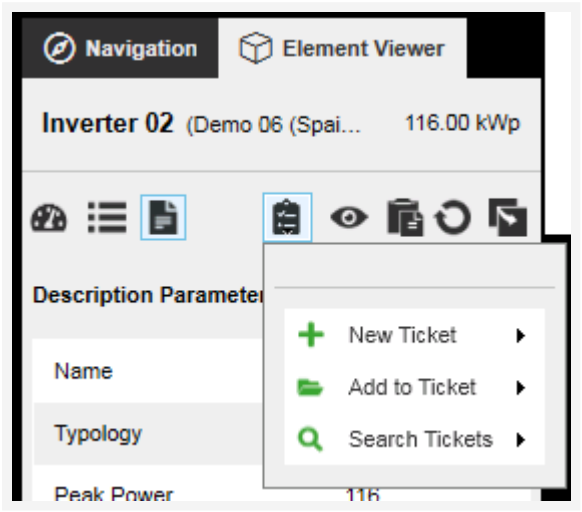
Create data correction tickets from elements

To create a data correction ticket from an element, follow these steps:

1. In the Element viewer module, click the icon.

   Result: The **Ticket** menu opens:

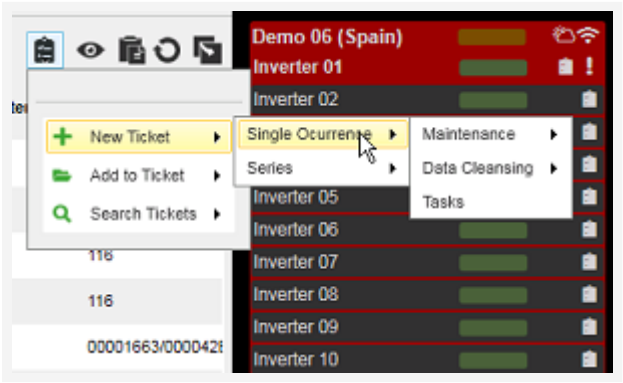
   ### Ticket menu

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.

   Result: The ticket type panel opens:

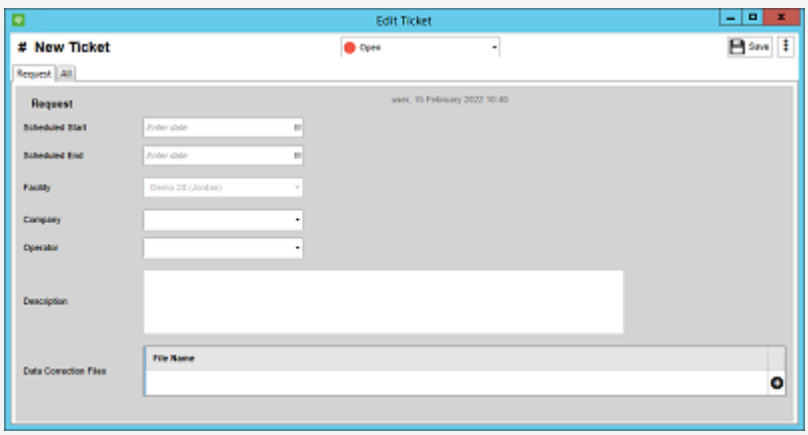
   ### Ticket types

3. In the menu, select **Data Cleansing**, then select **GPM Data Correction**.

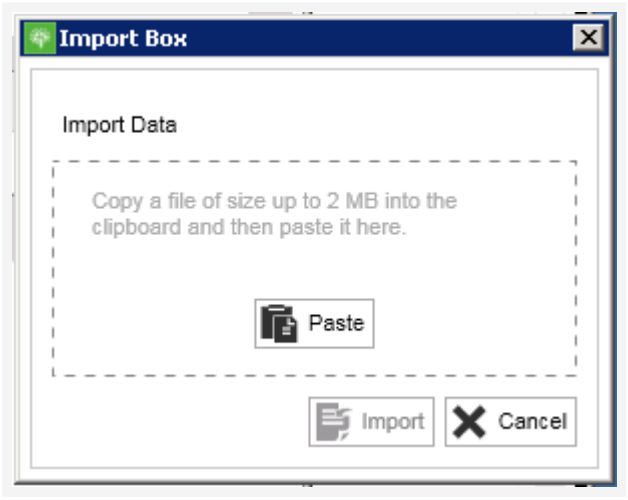
Result: The Edit ticket dialog appears:

Edit ticket dialog (Data cleansing)



4. In the Request section, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. *Description*: enter a description in the text input field.
5. In the Data Correction Files section, click the ⊕ icon to open the Import Box and import an XLS file from which to add the corrected data:

Import Box



ⓘ NOTE: You can only import XLS files.

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

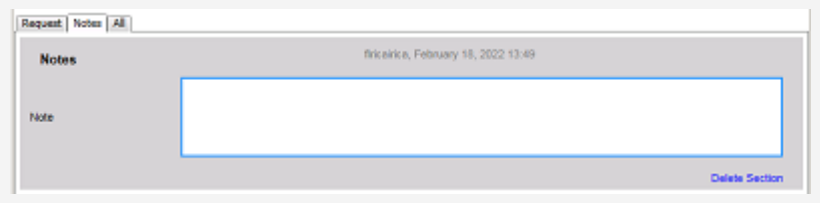
Result: The file is imported to the ticket.

6. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition



b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

7. Click **Save**.

Result

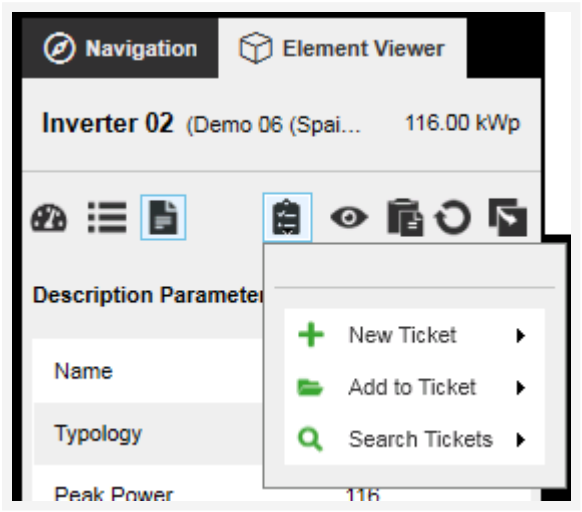
The ticket is created and assigned to the operator you selected.

Create task tickets from elements

To create a task ticket from an element, follow these steps:

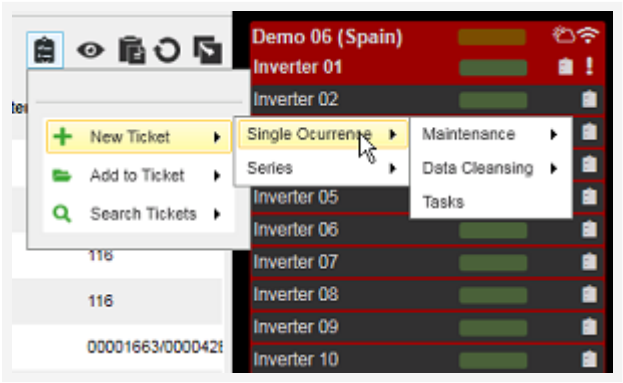
1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

Ticket menu



2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.
   Result: The ticket type panel opens:

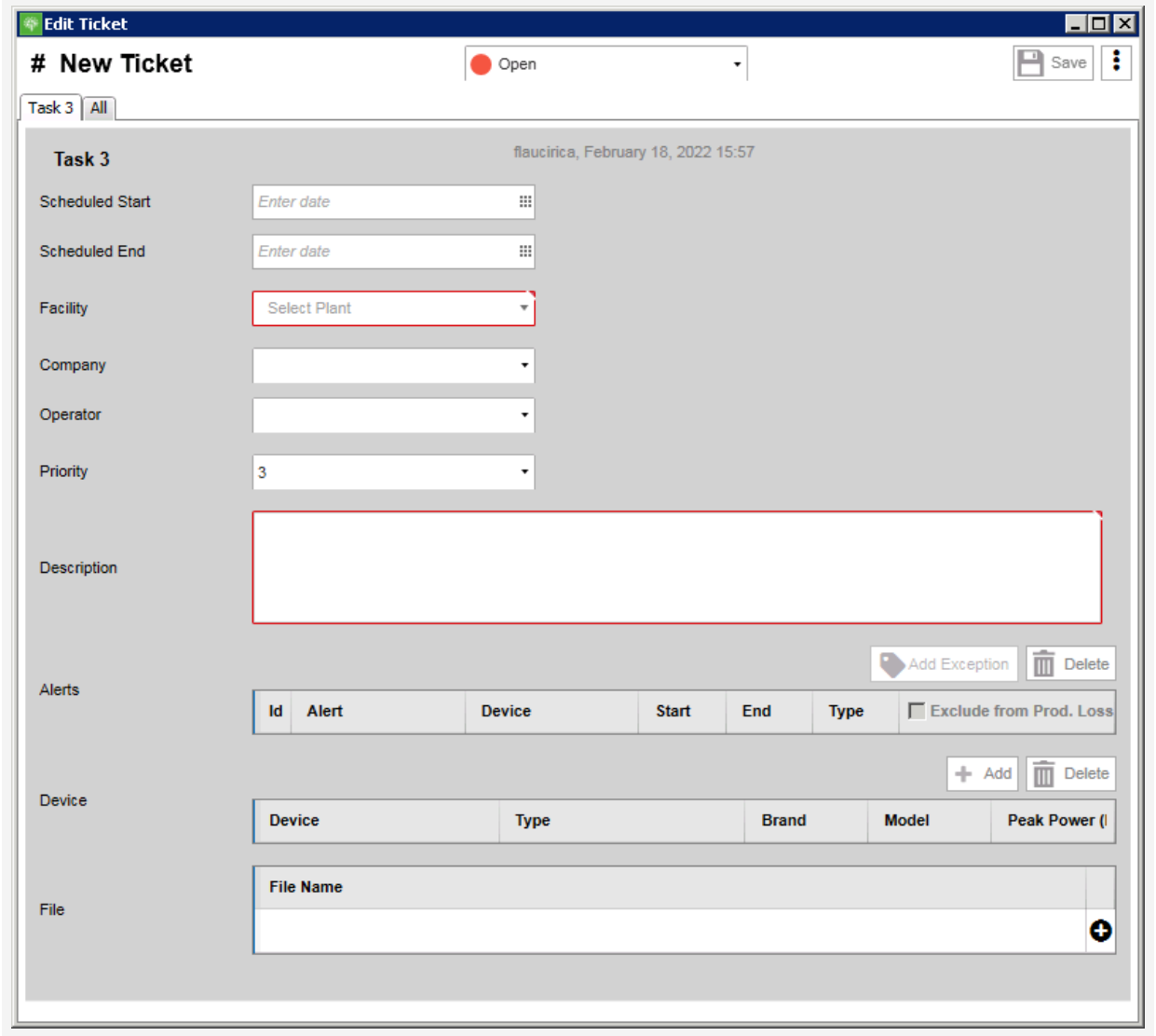
Ticket types



3. In the menu, select **Task**, then select the **GPM Default** template.

Result: The **Edit** ticket dialog appears:

Edit ticket dialog (task)



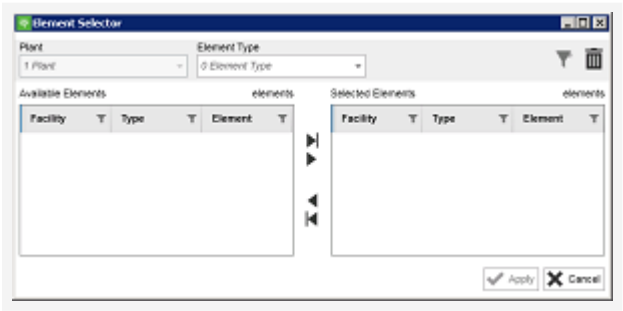
4. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

5 (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c Click **Apply**.
Result: The element is added to the list.

6 (Optional) In the File section, click the ⊕ icon to open the Import Box and import a file.

Import box

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

Result: The file is imported to the ticket.

7. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition

b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

8. Click **Save**.

Result

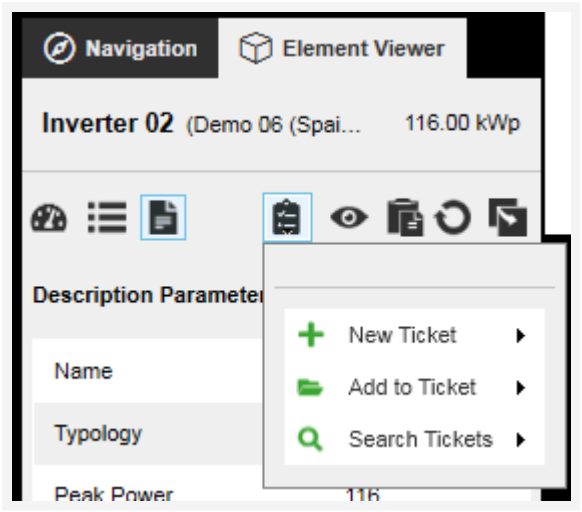
The ticket is created and assigned to the operator you selected.

Add elements to existing tickets

To add an element to an existing ticket, follow these steps:

1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

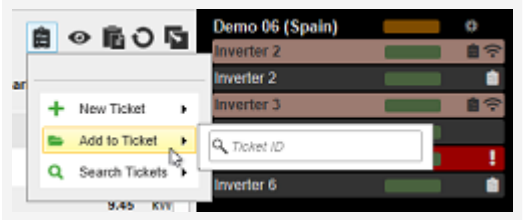
   ### Ticket menu

2. Hover over **Add to Ticket** and enter the *Ticket ID* of an existing ticket (for example, "1543549623"), and press **Enter**.

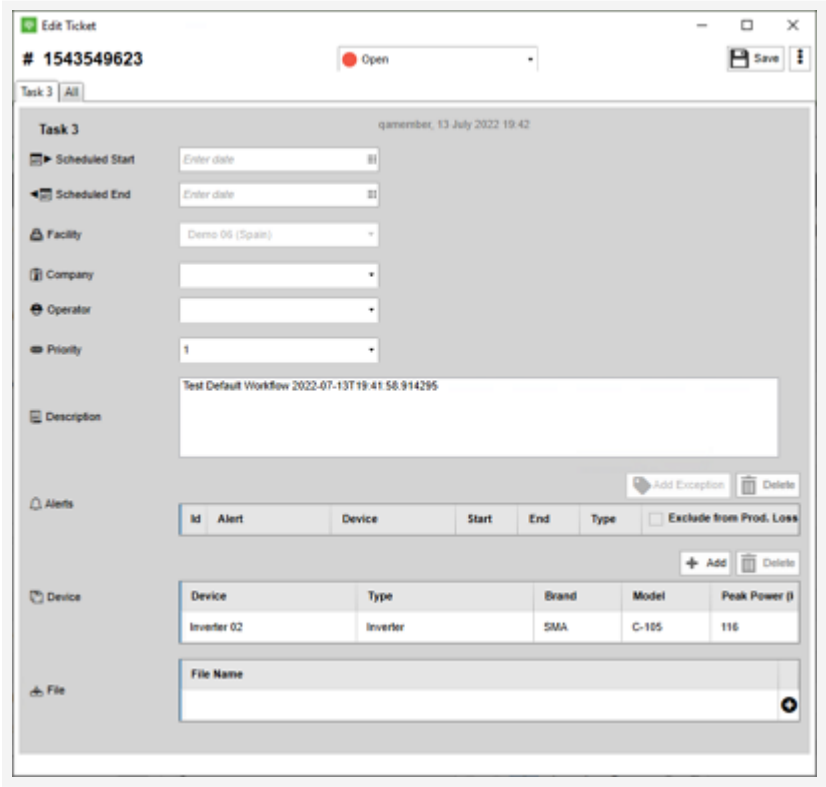
   > REMEMBER: You must enter the ID of an existing ticket. If no ticket exists, follow the instructions to create a new ticket from the Element viewer.

   ### Ticket menu

Result: The **Edit ticket** dialog appears, displaying the selected element in the Element field:

Edit ticket dialog



3. (Optional) Edit any other fields of the ticket you want to change.
4. Click **Save**.

Result

The element is added to the ticket and any other changes are saved.

Content area

The Content Area is organized in tabs and gives you access to the main tools and features of GPM SCADA. Each tab gives you access to a specific module, which consists of specific frameworks that you can use to monitor and analyze all the information in your portfolio.

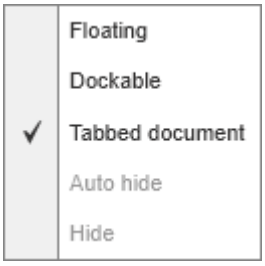
The main tabs in the content area are the following modules:

- Network Operating Center (NOC)
- Map
- Tickets
- Alarms

Content area

You can use drag-and-drop to change the order of the tabs. Also, you can right-click a tab and select one of the following display modes from the context menu.

Context menu



Option	Description
Floating	Click to undock the tab. You can use drag-and-drop to move it outside of the main window or to another screen.
Dockable	Click to undock the tab. To dock it again, you can use drag-and-drop and drop the tab on the icons indicating any side of the screen.
Tabbed document	When a tab is set to **Floating**, select this option to pin the tab to the Content Area.
Hide	When a tab is set to **Floating**, select this option to pin the tab to its default position in the Content Area.
Autohide	When a tab is docked, select this option to automatically hide the tab when you are not using it.

Alarms

Alarms are entities that contain a set of activation and a set of deactivation conditions. When an alarm is triggered, you receive a notification to inform you when there is an event that affects production in your site. You can monitor, analyze and manage alarms in the Alarms module.

Alarm triggers are the events that set off the alarm when a site or an element meets a given activation condition. You can perform tasks directly from the table of the Alarms module to address these conditions, such as creating a ticket and assigning it to an operator. The system deactivates the alarm when the deactivation conditions are met. It is also possible to deactivate certain alarms manually, depending on its configuration. For more information about activation

and deactivation conditions, see the article on alarm triggers.

The Alarms module allows you to perform several actions, depending on your role and the needs of your organization. For more information, see the section on working with alarms.

Alarm triggers

A trigger is the event that sets off the alarm when a site or an element meets a set of activation conditions. Each type of alarm has its own activation conditions. To deactivate the alarm, the element must meet the deactivation conditions defined for that trigger.

The system checks specific values for every element in a site periodically. When one of these values meets the activation conditions, the alarm trigger activates the alarm and notifies you in different screens of the user interface (for example, in the NOC module). In the Alarms module, you can see detailed information to interpret the activation and deactivation conditions of the alarm.

Activation conditions

Activation conditions are sets of values that trigger an alarm. These values can be specific (for example, "Status=0") or any number above or below a specific threshold (for example, "Power < 10").

An activation condition can have one or several values, depending on the parameters of a specific technology type, site or element. For instance, in a solar site, an inverter may trigger an alarm when its power output is below a specific threshold in relation to the irradiance (for example, "Power < 10" and "Irradiance > 3"). The relative nature of these thresholds prevents the system from incorrectly triggering an alarm for a solar panel at night, when the power output is expected to be zero. In wind sites, the same may happen when the generator of a wind turbine is generating less power than expected at a certain wind speed (for example, "Power < 10" and "Wind speed > 3").

> ⓘ NOTE: Some alarm types do not have activation conditions. For example, "Datalogger without communication" and "Plant communication stop."

Deactivation conditions

The system deactivates an alarm after the element meets the deactivation conditions. For example, a communication-type alarm may trigger when the system pings an element and does not receive an answer after a specific period of time. The alarm will be deactivated when the communications resume.

When an alarm does not have an explicit deactivation condition explicitly defined for it, the deactivation condition is the opposite of the activation condition. For example, if the activation condition for an alarm for a generator in a wind turbine is "Power <= 0", the deactivation condition will be "Power = 0".

Work with alarms

Alarms are mainly useful for two tasks: analyzing production and operations and management (O&M).

Analyze alarms

To analyze production, the main resource is the Alarm Information Window in the Alarms module. Here, you can assess how the condition that triggers an alarm impacts production.

The main analysis tasks for alarms are:

- View detailed information for alarms
- Interpret activation and deactivation conditions
- Apply filters to the alarms table
  - Basic filters: time period, plant and keywords.
  - Advanced filters: alarm ID, status, type, severity, and more.
- Export alarms from different areas of GPM SCADA:
  - Export data from the alarms grid
  - Generate reports for alarms

O&M tasks

O&M tasks allow you to address alarms by creating tickets, assigning them to operators and managing the status of an alarm. You can use the context menu of an alarm to directly access the available actions. Some of these are also available through other modules (for example, Tickets).

The main O&M tasks for alarms are:

- Assign alarms to operators

  > ⓘ NOTE: This option is not available when the alarm is already assigned to a ticket, or when there is no operator associated to the plant where the alarm is triggered.

- Ticket-related tasks
  - Create new tickets for alarms:

    > ⓘ NOTE: These options are not available when you select multiple alarm triggers from different plants.

    - Maintenance
  - Add alarms to tickets
  - Search for tickets linked to alarms
- Add exceptions to alarms
- Manage alarms:
  - Put alarms on hold
  - Deactivate alarms
  - Delete alarms

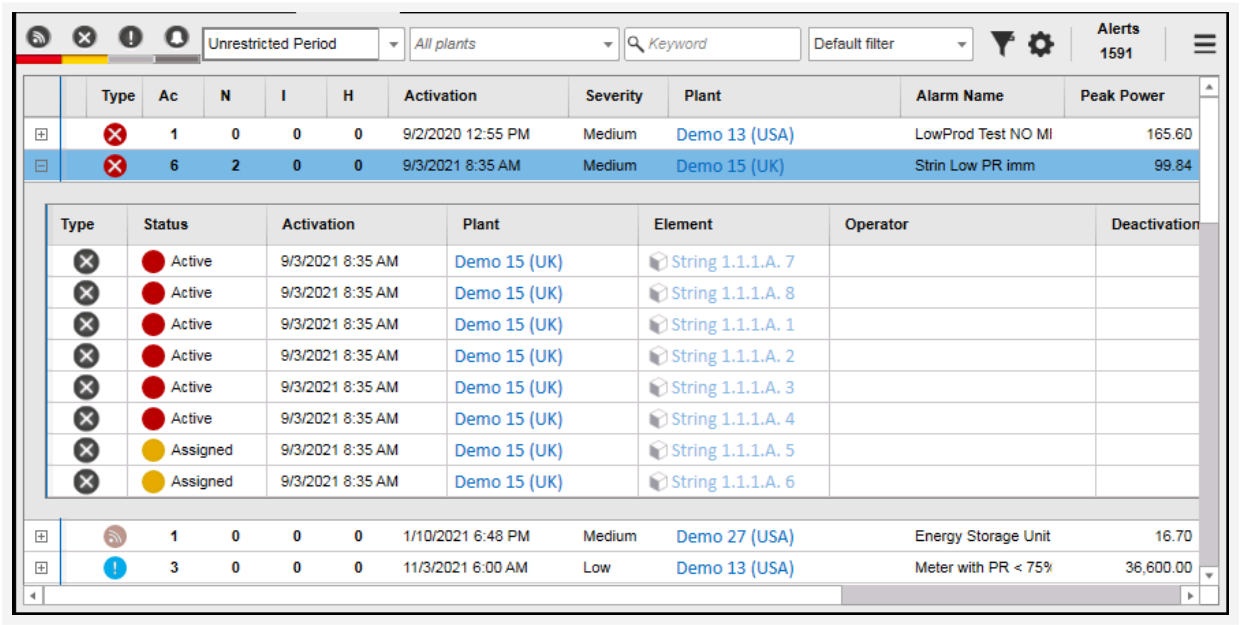
Assign alarms to operators

To assign an alarm to an operator, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

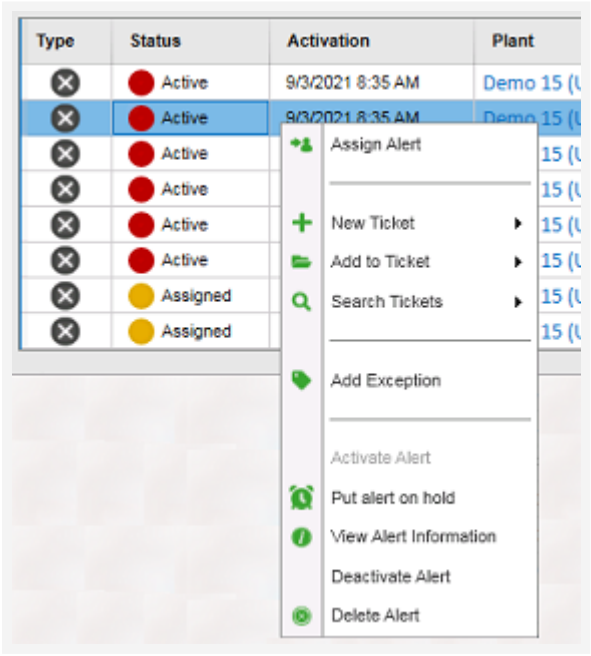


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the shift key to select multiple triggers. To select multiple triggers from different alarms, see the instructions to Ungroup alarm triggers.

Result: The context menu appears:

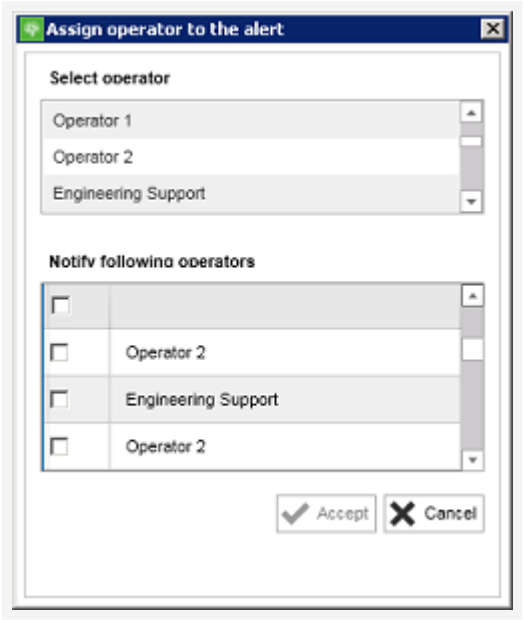
Alarm trigger context menu



3 Click **Assign Alarm**.

Result: The Assign alarm to operator dialog appears:

Assign alarm to operator dialog



4 In the **Select operator** section, click on the operator to whom you want to assign the alarm.

5 (Optional) Click the **Notify re assignment** checkbox to automatically notify the operator

of the assignment.

6. (Optional) In the **Notify following operators** section, select additional operators to whom you want to send a notification regarding the assignment.
7. Click **Accept**.

Result

The alarm is assigned to the operator.

> ⓘ NOTE: You can re-assign or unassign an alarm that has been assigned to an operator.

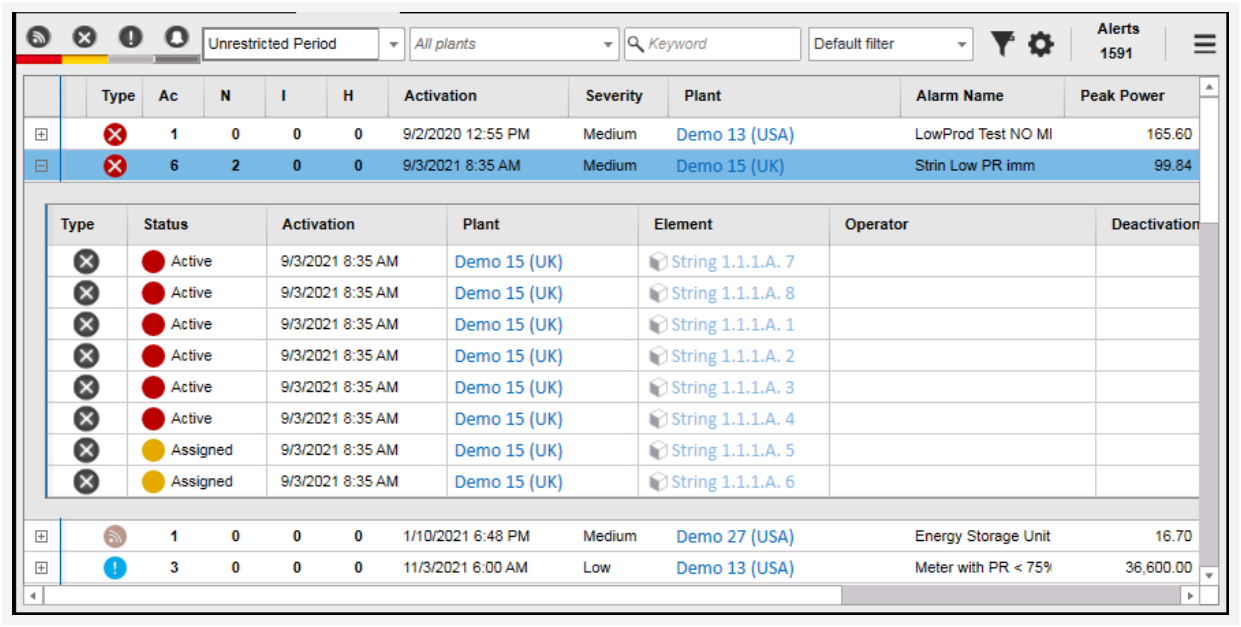
Add alarms to tickets

To add an alarm to a ticket, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

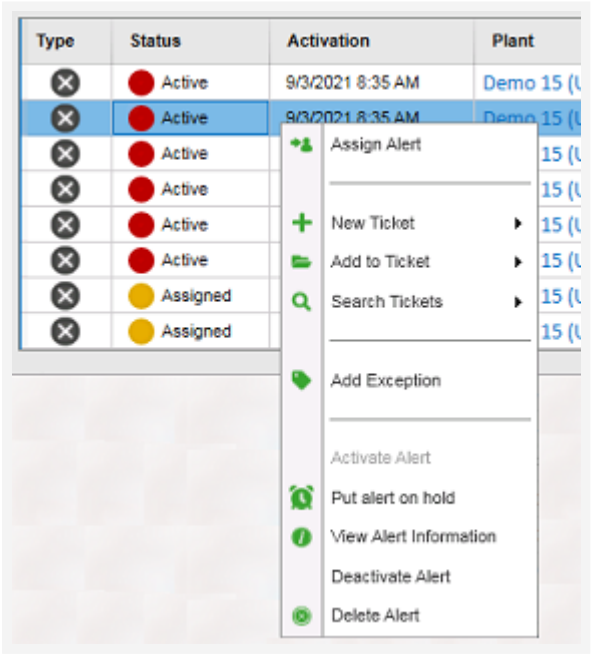


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the Ctrl key in your keyboard to select multiple triggers.

Result: The context menu appears:

Alarm trigger context menu

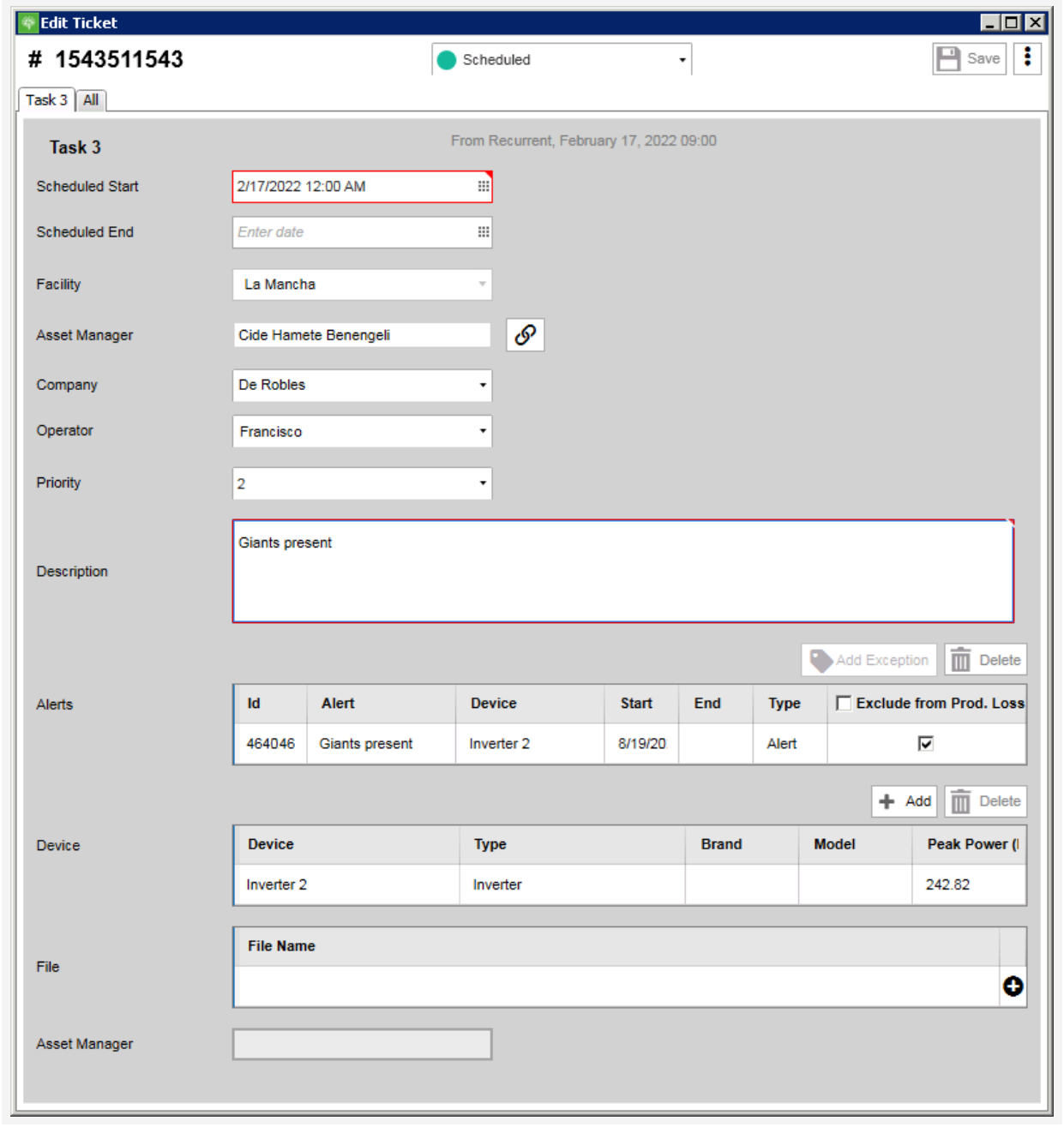


3 Hover over **Add to Ticket** and enter the *Ticket ID* in the search field, then press Enter.

> ⓘ NOTE: The ticket must belong to the same plant as the alarm.

Result: The Edit ticket dialog appears:

Edit ticket dialog



4 Click **Save**.

Result

The trigger is linked to the ticket.

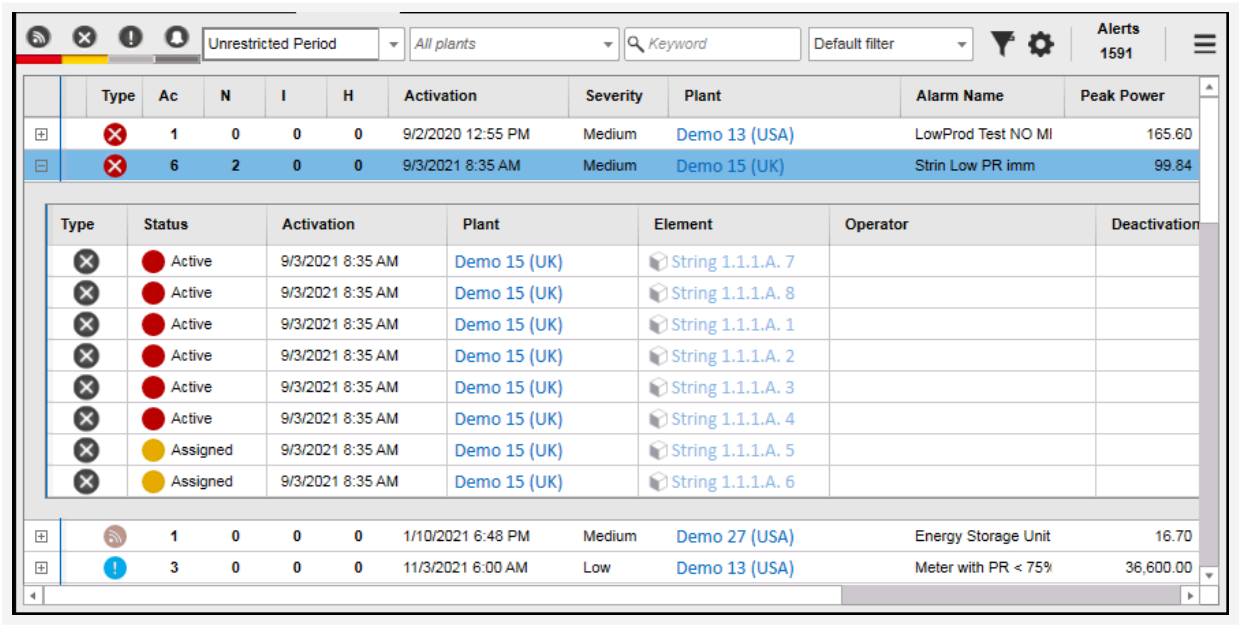
Create maintenance tickets for alarms

To create a maintenance ticket from an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

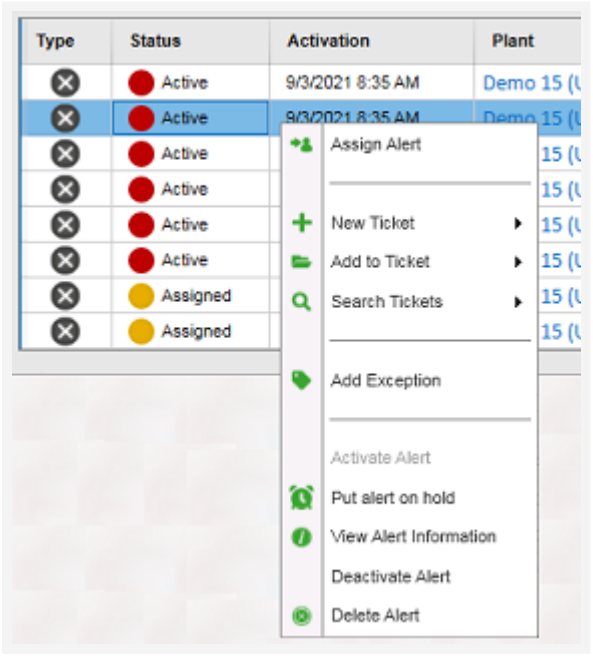


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the Ctrl key in your keyboard to select multiple triggers.

Result: The context menu appears:

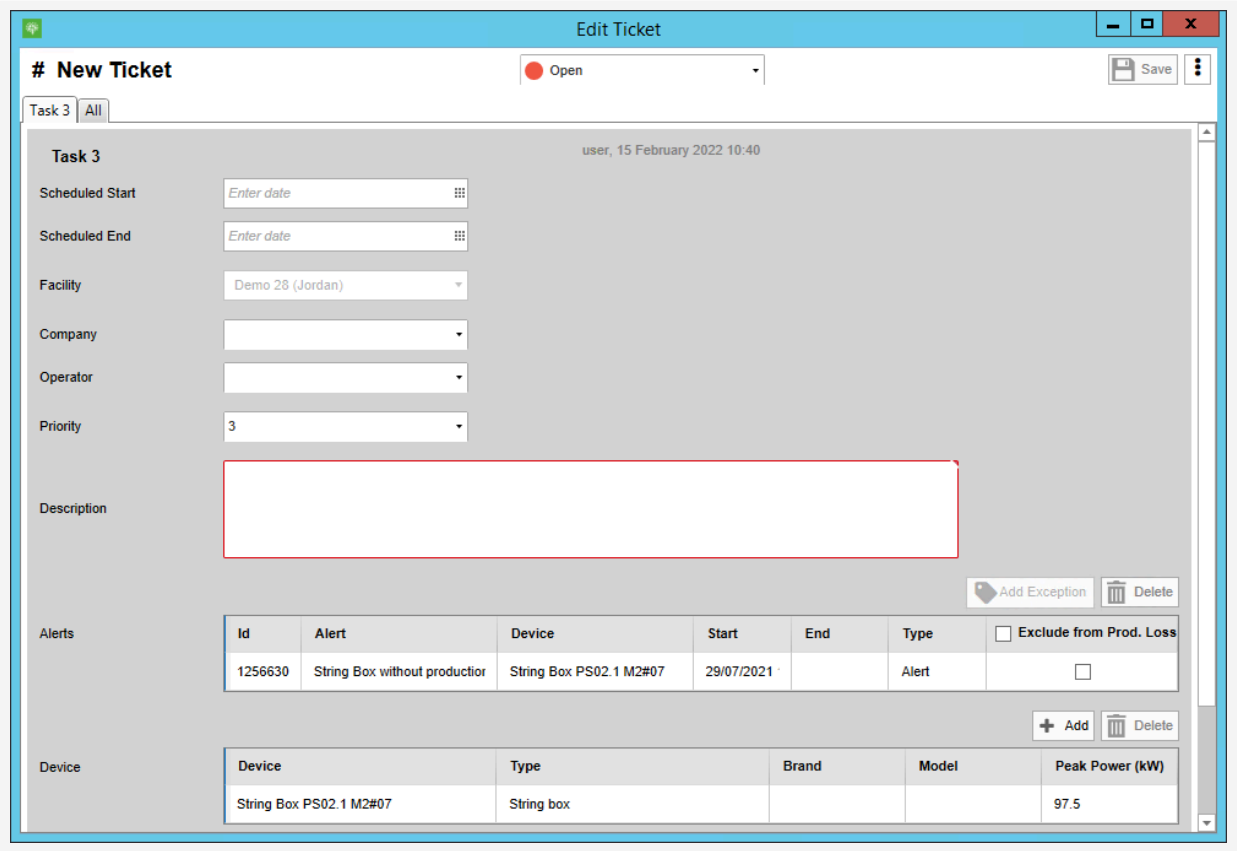
Alarm trigger context menu



3 Hover over **New Ticket** to expand the options panel, then hover over **Maintenance** to expand the templates panel. Select the template from which to create the ticket.

Result: The Edit Ticket dialog appears:

Edit Ticket dialog



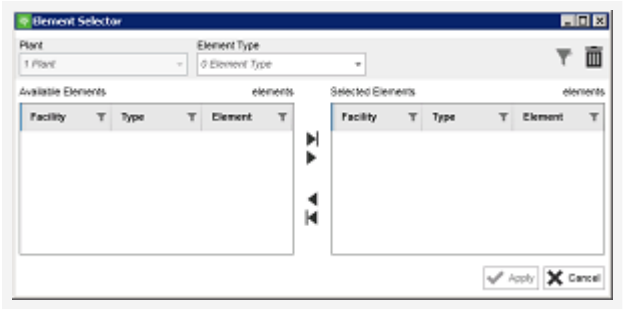
4. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. **Priority**: select a priority from the drop-down list.
      Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.
   g. *Description*: enter a description in the text input field.
   h. **Device**: if you have selected a plant, you can specify the elements to which the ticket applies
   i. (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information,

see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

5. (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a. Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the ▼ icon to narrow down the options available on the list.

b. In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c. Click **Apply**.
Result: The element is added to the list.

6. (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).
7. Click 💾 **Save**.

Result

The ticket is created and linked to the alarm.

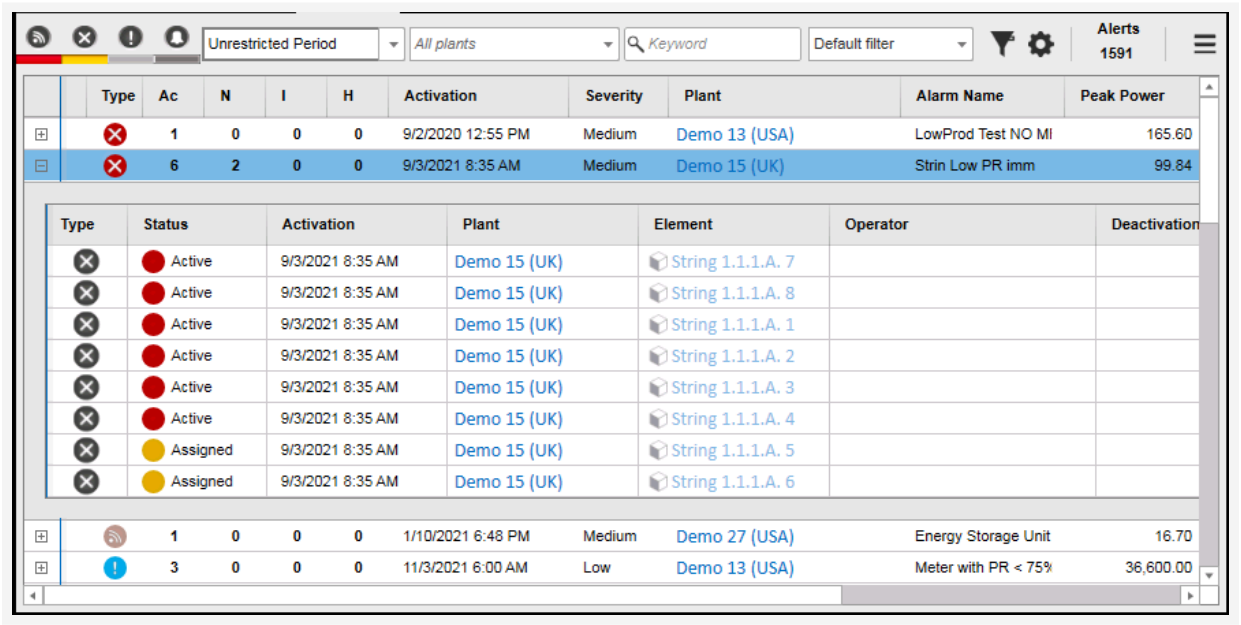
Search for tickets linked to alarms

To search for tickets linked to an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

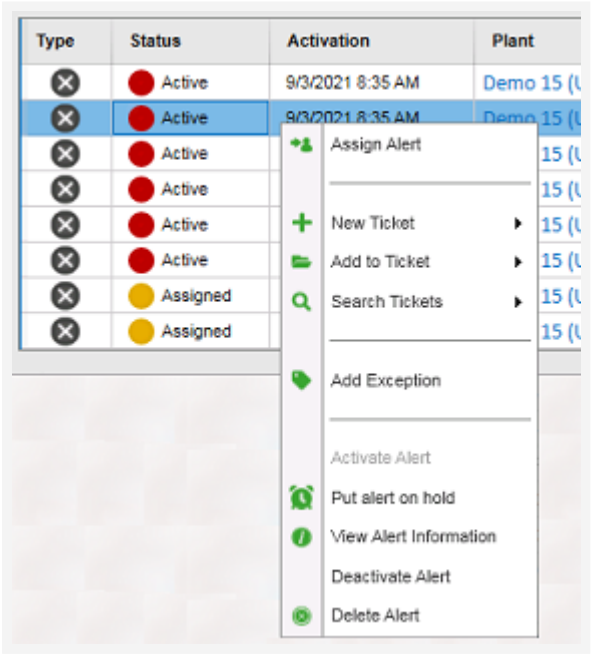


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the Ctrl key in your keyboard to select multiple triggers.

Result: The context menu appears:

Alarm trigger context menu

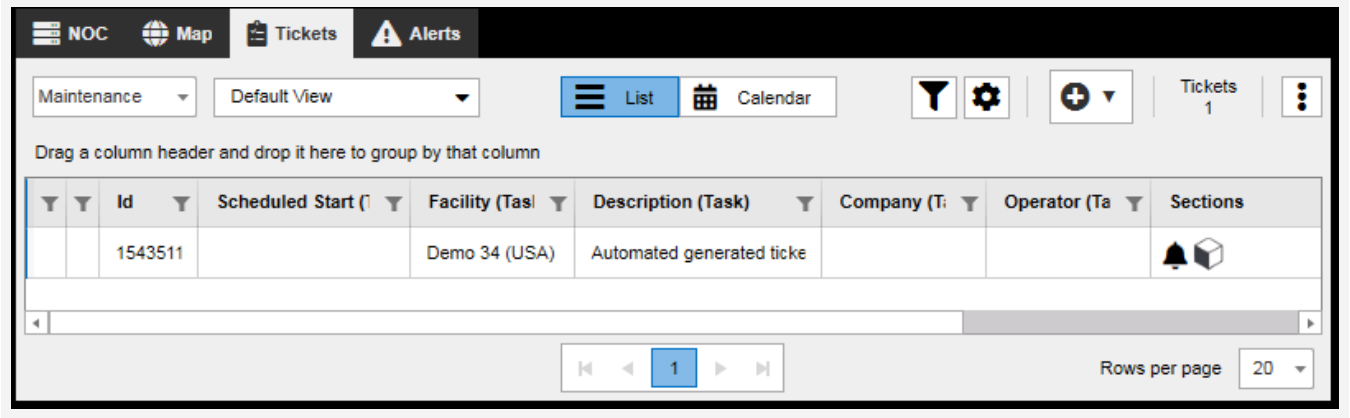


3 Click **Search tickets**.

Result

The Tickets module opens and displays the results:

Search results in Tickets module



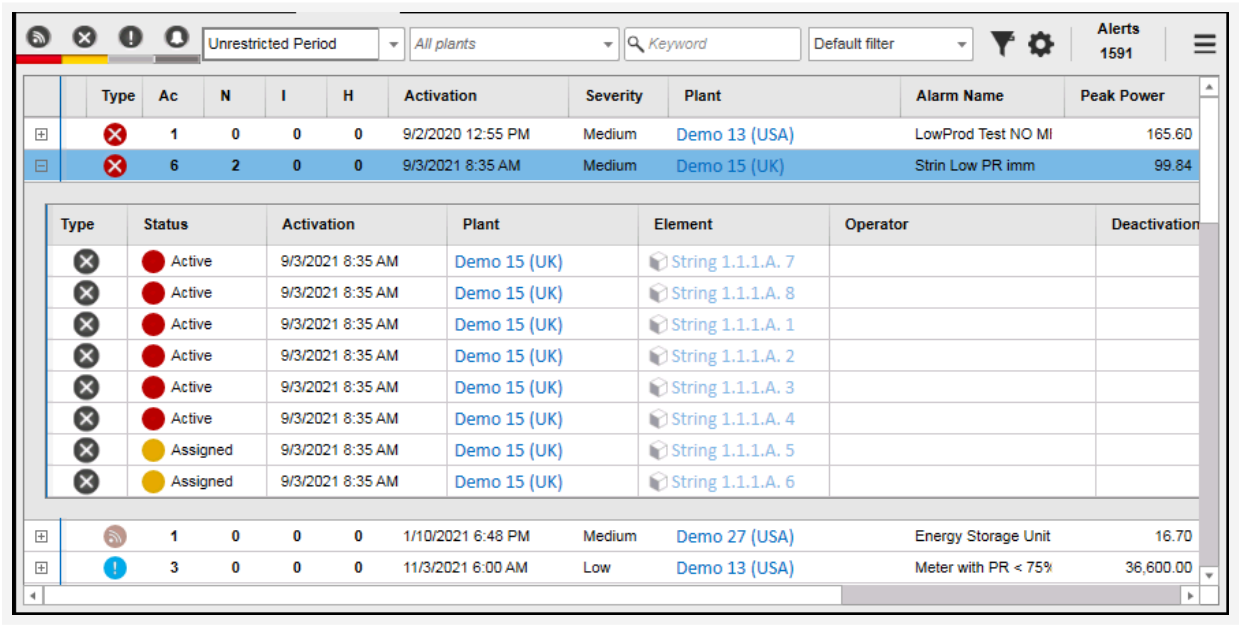
Add exceptions to alarms

To add an exception to an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

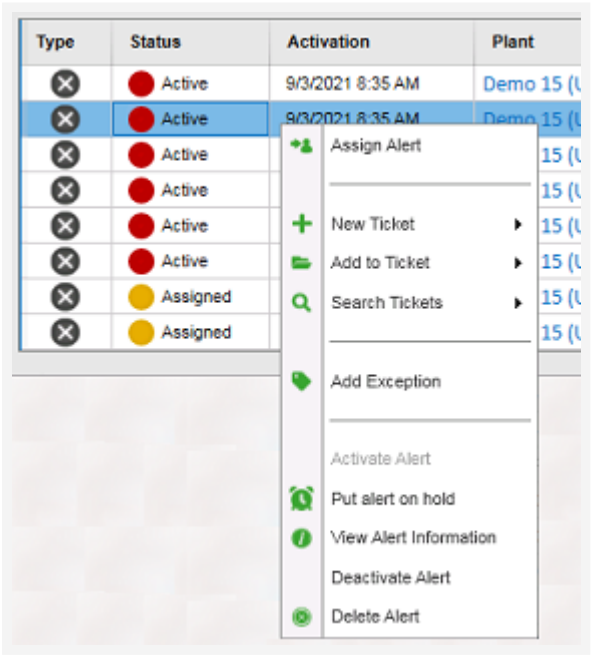
Expanded alarms



2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

Result: The context menu appears:

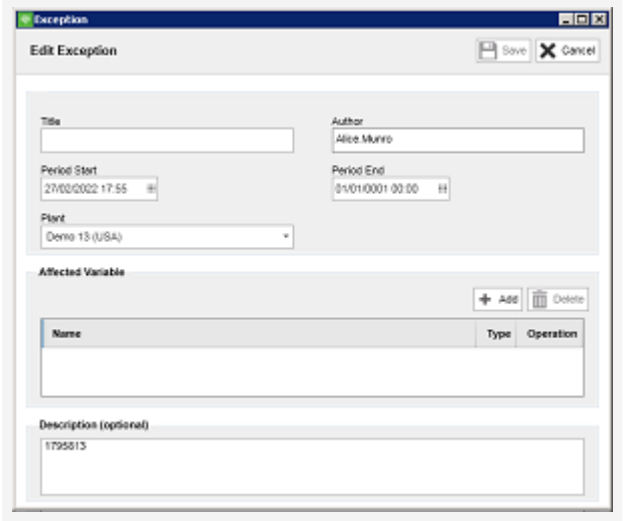
Alarm trigger context menu



3 Click **Add Exception**.

Result: The Edit Exception dialog appears:

Edit exception dialog



4 En the **Edit exception** dialog, enter the details for the exception:

> ⓘ NOTE: The *Author* field is automatically filled in with your username.

a. Enter a *Title* for the exception.

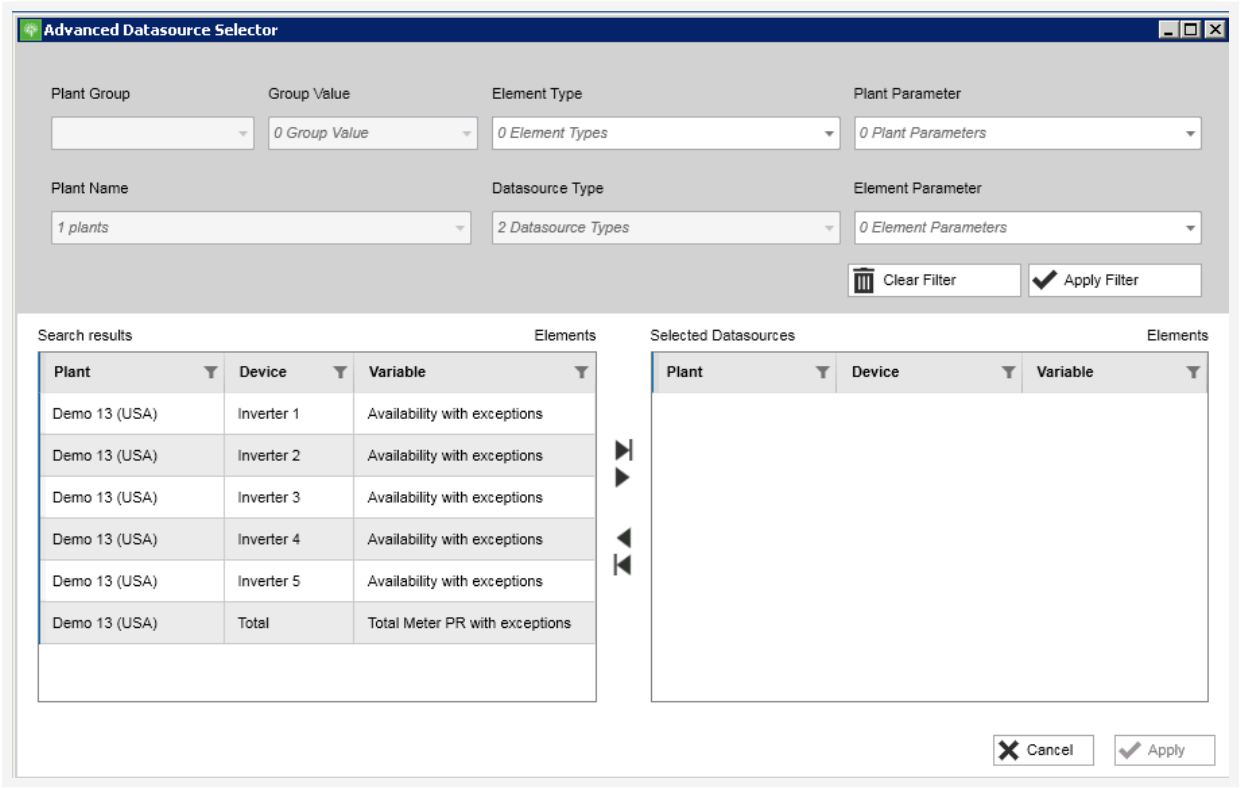
b. Click the **Period Start** menu and select the date and time at which the exception begins.

c. Click the **Period End** menu and select the date and time at which the exception ends.

5. In the **Affected Variable** section, click **+ Add**.

Result: The **Advanced Datasource Selector** dialog appears:

Advanced Datasource Selector



6. In the **Advanced Datasource Selector** dialog, select the variables you want to add to the exception:

a. In the Search Results panel, click the exceptions you want to add, then click the ▶ icon to move them to the **Selected Datasources** panel.

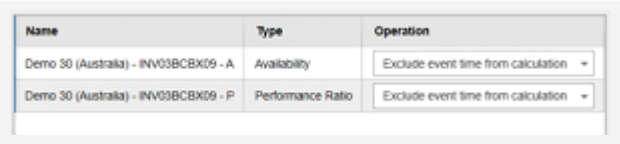
> TIP: Hold down the SHIFT key to select multiple variables.

OR: Click the ▶| icon to move all the available variables to the **Selected Datasources** panel.

b. Click ✓ **Apply**.

Result: The variables are added to the exception and an **Operations** menu appears for each variable:

Operations menu for variables



7. For each variable, click the Operation drop-down menu and select one of the available options:
   - **Exclude event time from calculation**: the calculation excludes data of the specified corresponding to the time period specified in Step 4.
   - **Force time as Available**: sets the status of the variables for the selected devices to "Available" for the time period specified in Step 4.

     ⓘ NOTE: This option is only available for Availability.
   - **Force time as Unavailable**: sets the status of the variables for the selected devices to "Unavailable" for the time period specified in Step 4.

     ⓘ NOTE: This option is only available for Availability.
8. (Optional) Enter a *Description* for the exception to include additional details.
9. Click **Save**.

Result

The exception is added to the alarm.

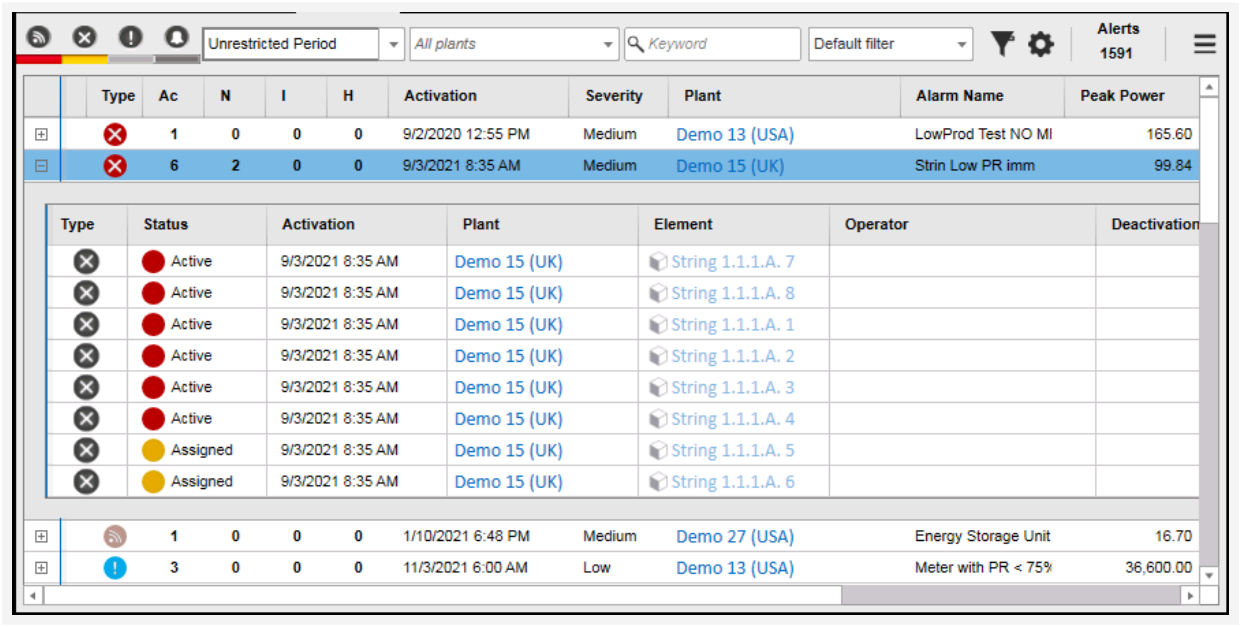
Put alarms on hold

To put an alarm on hold, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

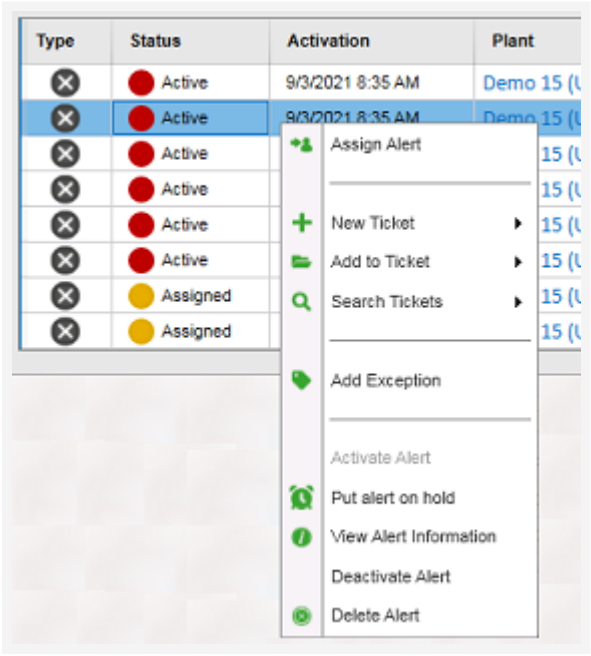


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the shift key to select multiple triggers. To select multiple triggers from different alarms, see the instructions to Ungroup alarm triggers.

Result: The context menu appears:

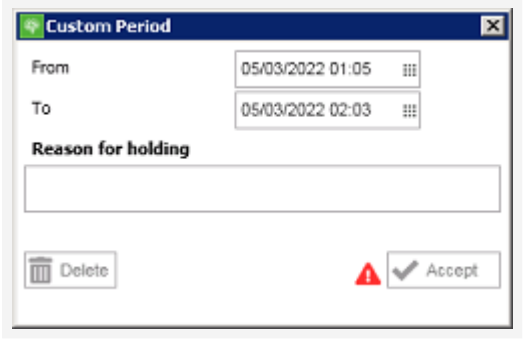
Alarm trigger context menu



3 Click **Put alarm on hold**.

Result: The **Custom Period** dialog appears:

Custom Period dialog



4 In the Custom Period dialog, enter the details for the alarm status:

a Click the **From** menu and select the date and time on which the hold status starts.

b Click the **To** menu and select the date and time on which the hold status ends.

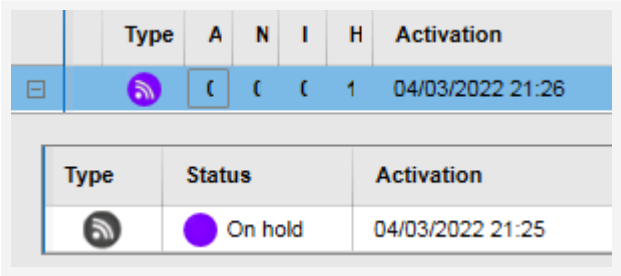
c Enter a *Reason for holding*.

d Click ✓ **Accept**.

Result

The alarm status changes to "On hold":

Alarm on hold



ⓘ NOTE: You can edit the hold information by opening the context menu for the alarm and selecting **Edit hold information**.

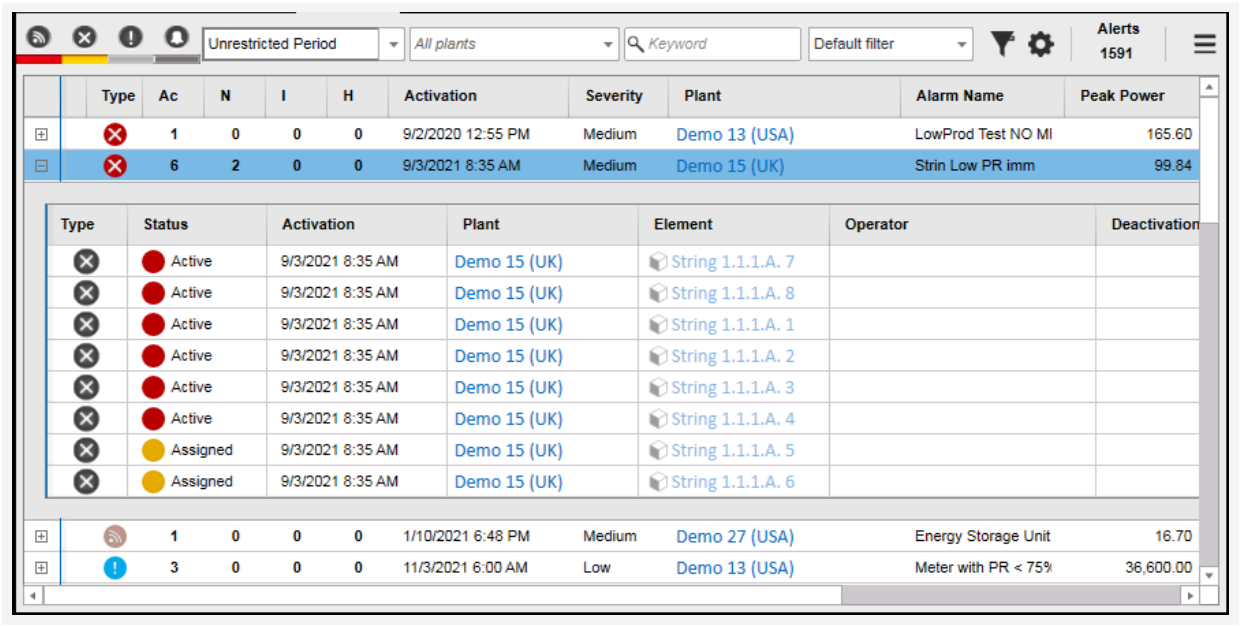
View detailed information for alarms

To view the information for an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

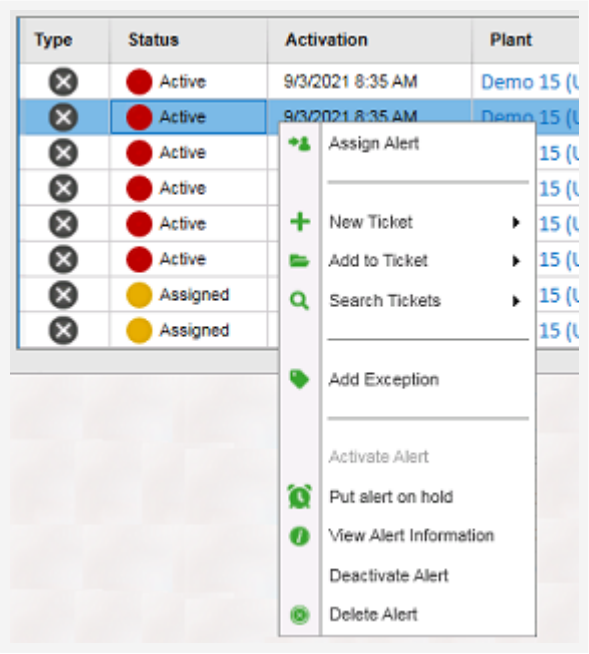
Expanded alarms



2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

Result: The context menu appears:

Alarm trigger context menu

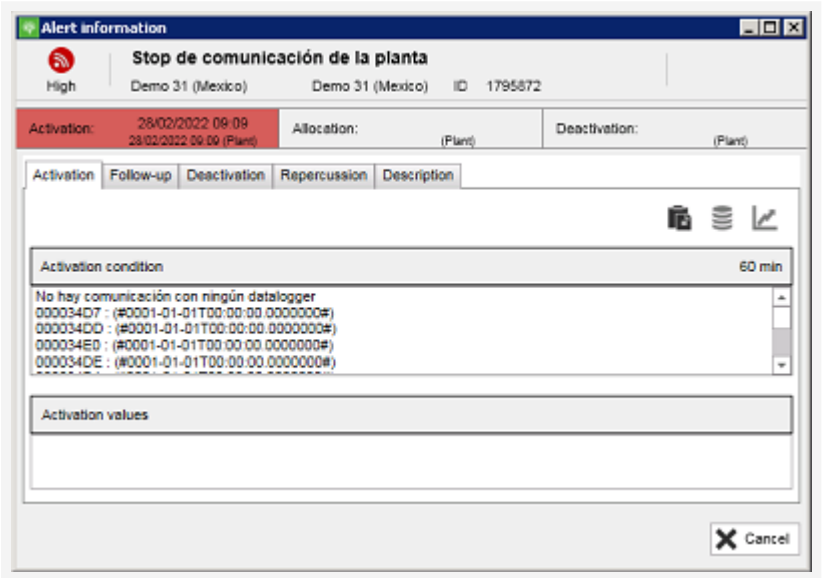


3 Click **View Alarm Information**.

Result

The Alarm information window appears:

Alarm information window



Deactivate alarms manually

Before you begin

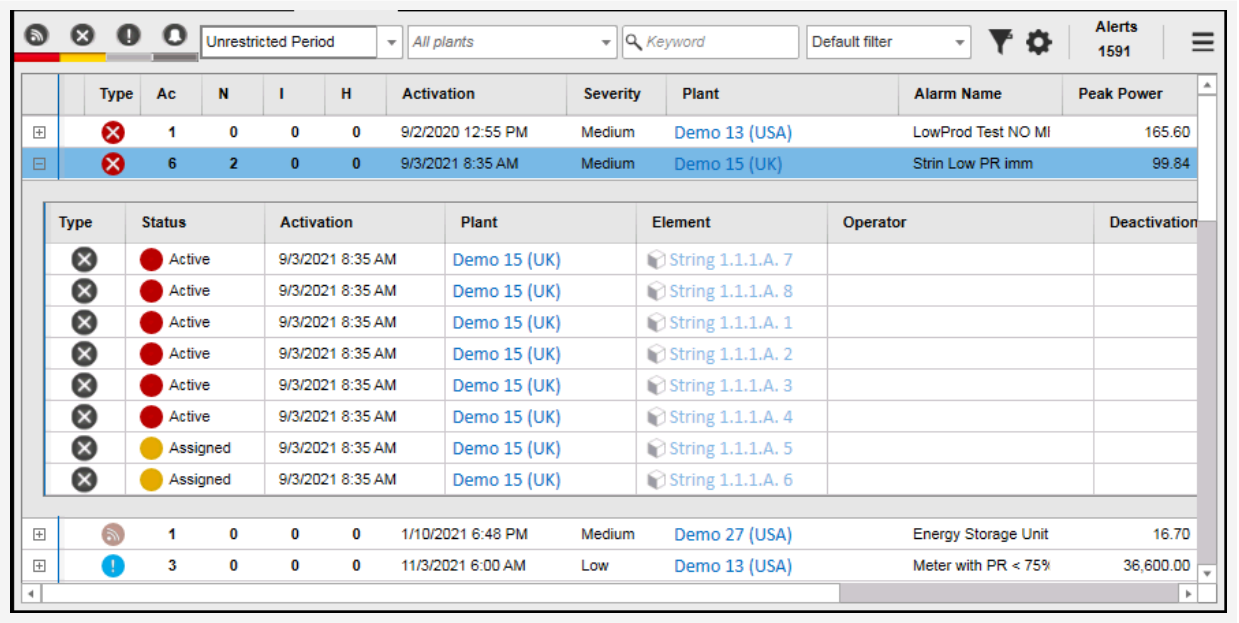
This task requires an administrator password for security validation.

To deactivate an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

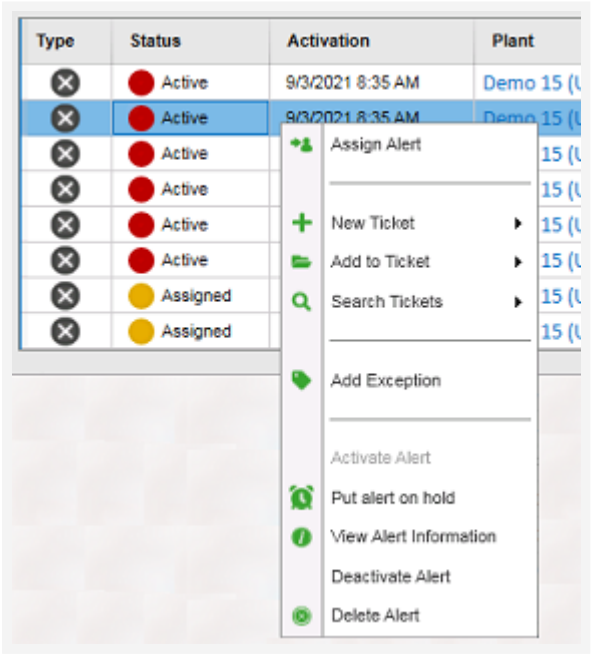


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the shift key to select multiple triggers. To select multiple triggers from different alarms, see the instructions to Ungroup alarm triggers.

Result: The context menu appears:

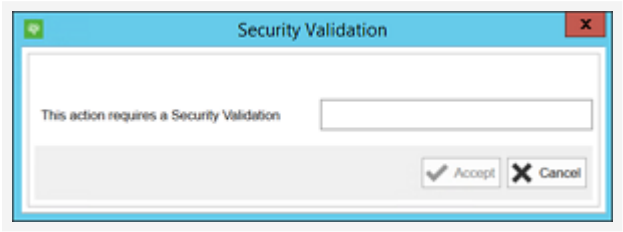
Alarm trigger context menu



3 Click **Deactivate alarm**.

Result: The Security Validation dialog appears:

Security validation



4 Enter the password and click ✓ **Accept**.

Result

The alarm is deactivated.

> ⓘ NOTE: If the activation condition that triggered the alarm is still present after a predefined period of time, a new alarm will be triggered and you will receive a notification with a new Alarm trigger ID and timestamp.

Delete alarms

Before you begin

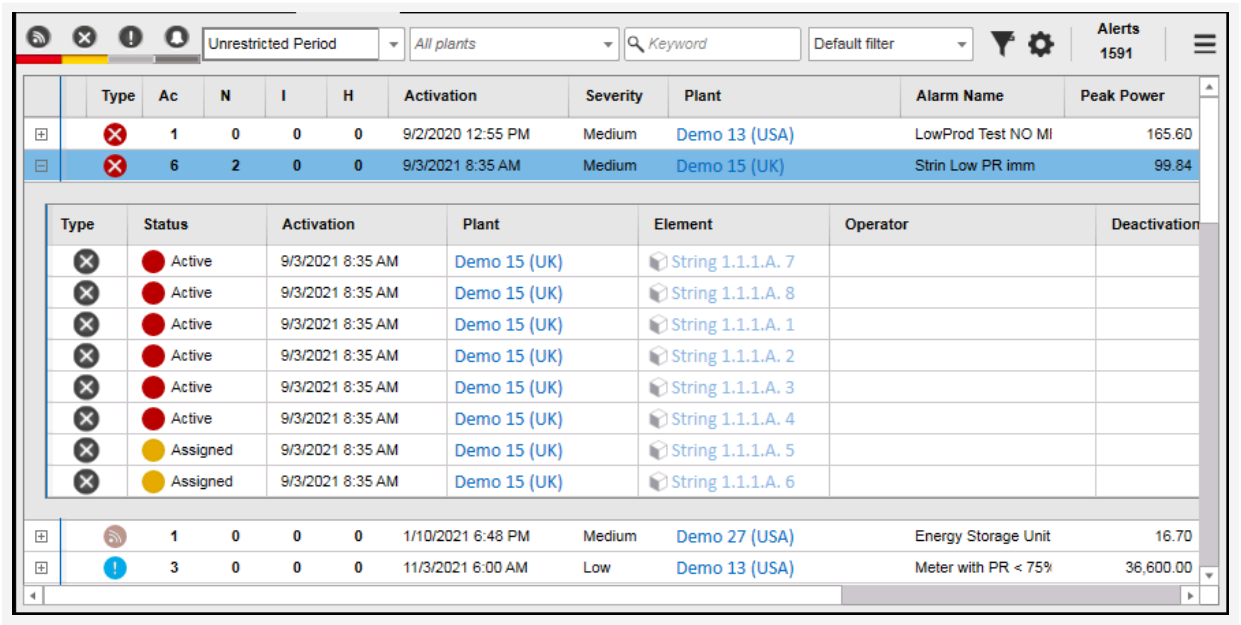
This task requires an administrator password.

To delete an alarm, follow these steps:

1. In the Alarms table of the Alarms module, click the **+** sign next to the group of alarm triggers you want to expand.

   Result: The collapsed list of triggers expands:

Expanded alarms

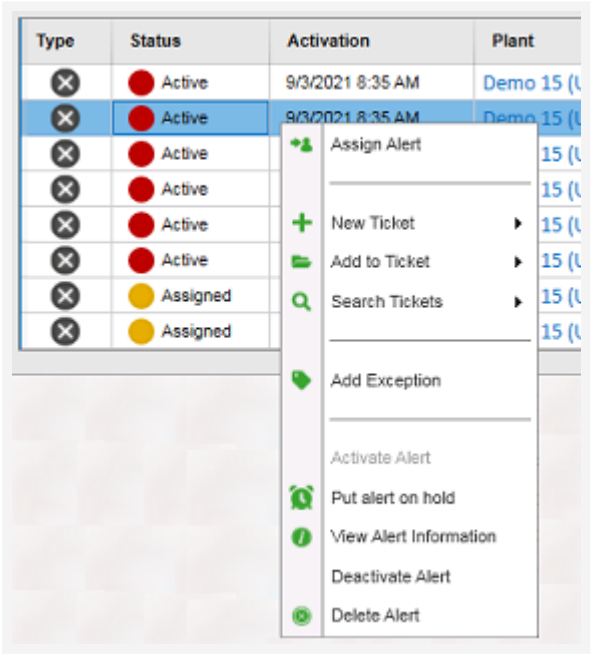


2. Click the alarm trigger you want to address to select it, then right-click it to open the context menu.

> TIP: Hold down the shift key to select multiple triggers. To select multiple triggers from different alarms, see the instructions to Ungroup alarm triggers.

Result: The context menu appears:

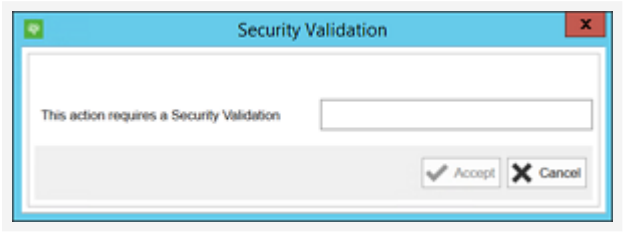
Alarm trigger context menu



3 Click **Delete alarm**.

Result: The Security Validation dialog appears:

Security validation



4 Enter the password and click ✓ **Accept**.

Result

The alarm is deleted.

Commands

Commands and command sequences allow you to take different actions to control the productivity of your plant and its physical devices. You can perform all the tasks related to

commands in the Commands Control and the Power Plant Control (PPC) modules.

From the PPC, you can perform the following tasks:

- Send set point values: control the set points of your plant.
- Schedule commands: configure a command to automatically send it to your plant at regular intervals.
- Send command sequences: control complex sets of commands. For more information, see the Commands sequences section.

> ⚠ CAUTION: Use extreme caution and follow all the safety procedures before performing any action related to commands. These actions directly impact the plant.

Command sequences

Command sequences are sets of commands that you can send to your plant in a pre-defined order. You can also define launch conditions for each sequence.

> ⓘ NOTE: To configure command sequences, contact your GPM representative.

Communications with physical devices occur through a proprietary GPM protocol that digests the data received from devices and makes them usable for the application.

You can send previously configured command sequences from the following modules in the user interface:

- Commands Control
- Vectorial Layout
- Element Viewer

> ⓘ NOTE: Command sequences con only be sent to modify parameters that have "write" permissions.

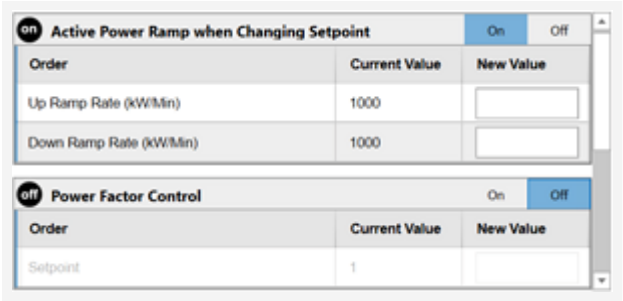
Send set point values to plants

To send a set point value to a plant, follow these steps:

> ⚠ CAUTION: Use extreme caution and follow all the safety procedures before performing any action from the PPC module. These actions directly impact the plant.

1. On the **Set Point Controls** panel of the Power Plant Control module, identify the set point that you want to control and click **On** to enable it.

   ### Set Point Controls panel

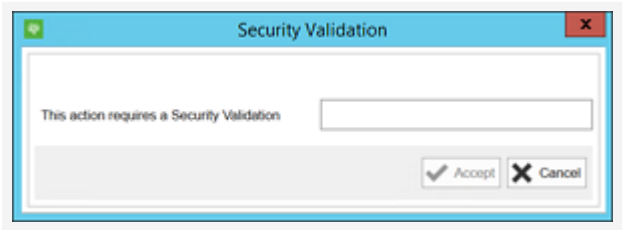
2. In the *New Value* field, enter the new value.

   > ⓘ NOTE: The application automatically suggests the last value entered.

3. Click **APPLY CHANGES**.

   Result: The **Security Validation** dialog appears:

   ### Security Validation dialog

4. On the **Security Validation** dialog, enter the administrator password and click **Accept**.

   > ⓘ NOTE: If you do not have the administrator password, contact your GPM representative

Result

The set point value is sent to the plant.

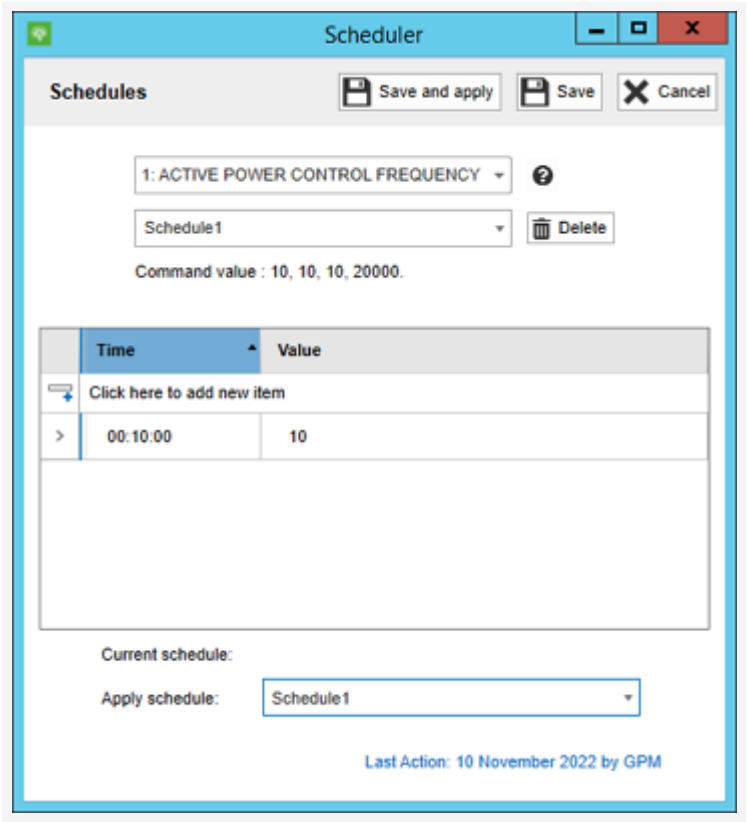
Create scheduled commands

You can create a scheduled command from the step 1 and the Power Plant Control Module.

> ⓘ NOTE: You can only schedule one modifiable parameter for each command.

1. In the **Actions** column of the Commands Control Module, click the icon.
   Result: The **Scheduler** dialog appears:

Scheduler dialog



2. In the Scheduler dialog, enter the information to create the command:
   a. **Parameter**: select one of the parameters configured for the command.
   b. **Scheduled commands**: click **+ New Schedule** to create a new command or select a preexisting command to edit it.
   c. **Command scheduled table**: click the **+** icon on the left-hand column to add the time and value for the command execution within the 24-hour period.
   d. Current schedule: displays the scheduled command that applies currently.
   e. **Apply schedule**: select the scheduled command to apply.

3 Click **Save**, or click **Save and Apply** to automatically execute the command.

> ⓘ NOTE: It is not possible to save commands with values that are not valid. This usually applies to values that are out of range (according to the Metacommand control). If you try to save commands with values that are not valid, the system notifies you and prompts you to correct them before completing the process.

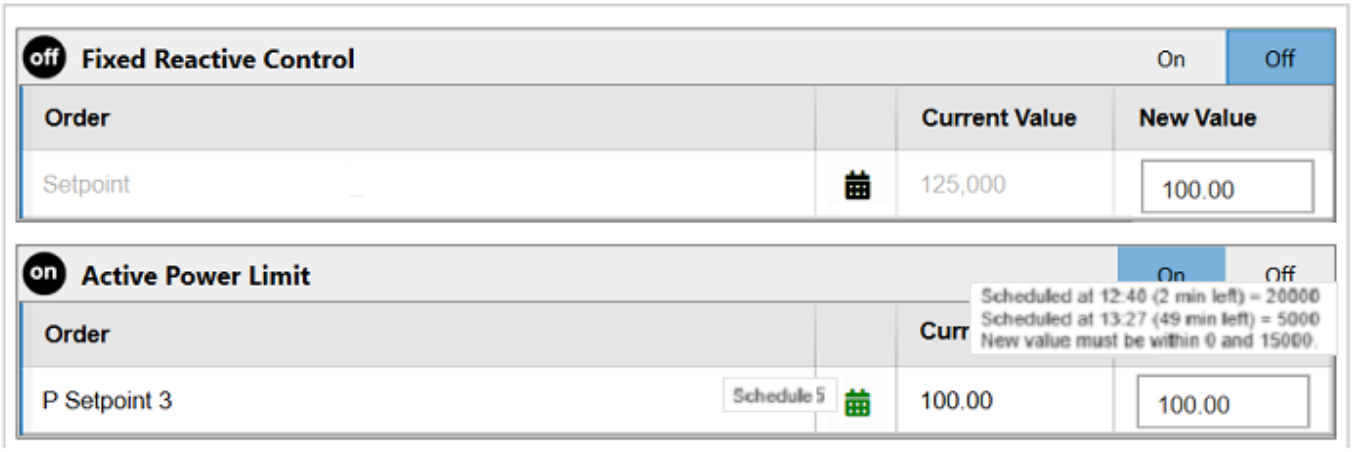
Result: The **Modify Metacommand** dialog appears.

4 In the **Modify Metacommand** dialog, enter your administrator password and click **Apply**.

Result

The command is created and scheduled for execution. In the PPC module, the icon will appear in the related command parameter.

Scheduled command icon



The icon displays the status of the scheduled command in two colors:

- Green: the scheduled command is currently active.
- Black: the scheduled command is currently inactive or no schedule has been configured.

You can obtain more information on the active scheduled command by doing the following:

- Hover over the icon to view the currently active schedule.
- Hover over the *New Value* input box to access additional details.

If you want to view and edit the scheduled command, you can access the Figure 95. Scheduler dialog by selecting the icon.

> ⓘ NOTE: If the icon does not appear after creating a scheduled command, contact your GPM representative.

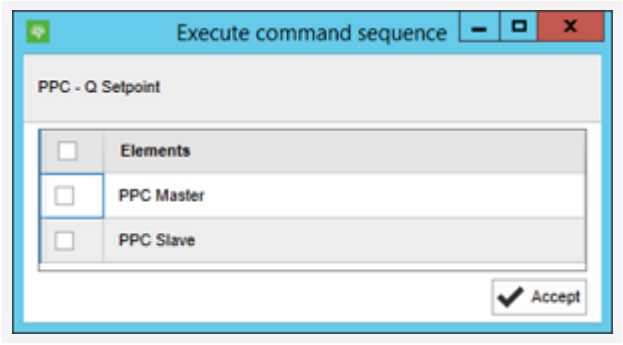
Send command sequences from the Commands Control module

To send a command sequence from the Commands Control module, follow these steps:

> ⚠ CAUTION: Use extreme caution and follow all safety procedures before following these steps. They have a direct impact on the plant's hardware.

1. In the **Actions** column of the command sequence, click the 👆 icon.
2. (Optional)) If your command affects multiple devices, on the **Execute Command Sequence** dialog, select the devices to which you want to send the command sequence and click **Accept**:

Execute Command Sequence dialog



3. On the **COMMAND EXECUTION** dialog, enter the administrator password and click **Accept**.

Result

The command sequence is sent to the selected elements and a confirmation message appears:

Command execution confirmation

To check if the command sequence is successfully sent, click **See Command Execution Status**. This takes you to the Command History tab of the Commands Control module and displays the history of the related command. If you want to close the dialog, click **Accept**.

Send command sequences from the Vectorial Layout module

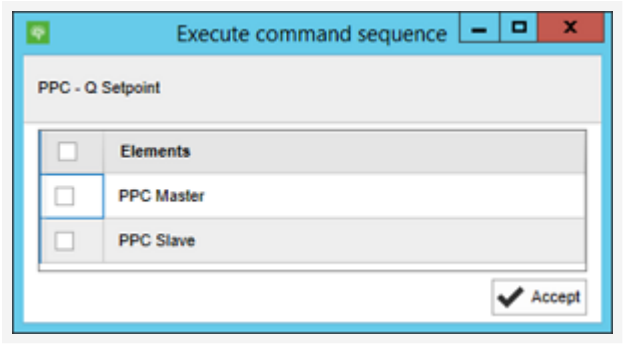
To send a command sequence from the Vectorial Layout module, follow these steps:

⚠ CAUTION: Use extreme caution and follow all safety procedures before following these steps. They have a direct impact on the plant's hardware.

ⓘ NOTE: The Vectorial Layout module can include buttons that act as shortcuts for commands. These buttons are customizable in column and shape, so they vary between user. In your configuration, the buttons will look the way you agreed with your GPM representative.

1. Click the button for the command you want to send.
2. (Optional)) If your command affects multiple devices, on the **Execute Command Sequence** dialog, select the devices to which you want to send the command sequence and click **Accept**:

   Execute Command Sequence dialog

3. On the **COMMAND EXECUTION** dialog, enter the administrator password and click **Accept**.

Result

The command sequence is sent to the selected elements and a confirmation message appears:

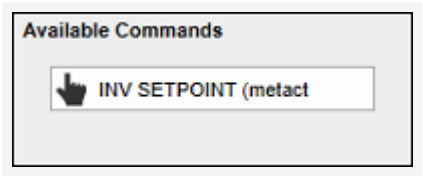
To check if the command sequence is successfully sent, click **See Command Execution Status**. This takes you to the Command History tab of the Commands Control module and displays the history of the related command. If you want to close the dialog, click **Accept**.

Send command sequences from the Element Viewer module

To send a command sequence from the Element Viewer module, follow these steps:

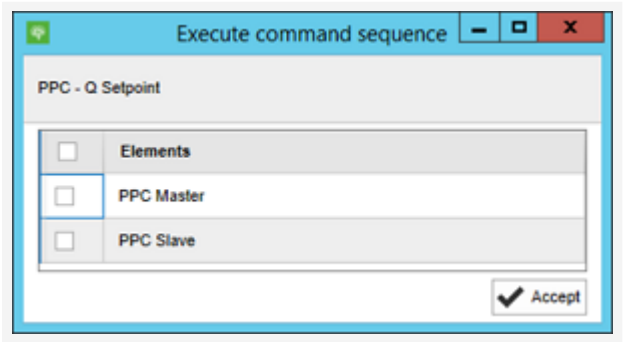
1. In the **Commands** tab of the Element Viewer module, click the command sequence that you want to send:

   Commands tab

2. (Optional)) If your command affects multiple devices, on the **Execute Command Sequence** dialog, select the devices to which you want to send the command sequence and click **Accept**:

   Execute Command Sequence dialog

3. On the **COMMAND EXECUTION** dialog, enter the administrator password and click **Accept**.

Result

The command sequence is sent to the selected elements and a confirmation message appears:

To check if the command sequence is successfully sent, click **See Command Execution Status**. This takes you to the Command History tab of the Commands Control module and displays the history of the related command. If you want to close the dialog, click **Accept**.

Entities

Entities are specific objects that contain information in the GPM system. They are the main sources of information to analyze, assess and manage portfolios, plants and elements.

Parameters

A parameter is a measurable factor that allows you to assess the performance or the status of your portfolio, a plant or an element within a plant. The values for parameters are usually numerical, but may sometimes be expressed as text (for example, "Offline" for the status of an element in a plant). These values for the data come from datasources.

Examples of parameters:

- Power
- Energy
- Performance Ratio (PR)
- Irradiation
- Wind speed

Descriptions

A description is metadata of a plant or an element within a plant. Descriptions usually have fixed values that are not necessarily numerical (for example, "Wind" is a value for "Technology type").

The flexibility of the GPM system allows you to use them as categories that you can modify manually (for example, the "Assigned Operator" of an alarm).

Examples of descriptions:

- Location (country or geolocation)
- Technology (wind, solar, storage)
- Element type (wind turbine, generator, inverter, weather station, etc.)

Datasources

A datasource is a source of data that enters parameter values from one or more elements into the system. The most basic datasources provide raw data (for example, irradiation, wind speed, and power) monitored directly from elements in the plant. Advanced datasources apply calculations to raw values, producing more complex data (for example, production ratios), and may be customized by GPM to meet the needs of your organization.

The datasource may input raw data, or process it beforehand, depending on its configuration.

Examples of datasources:

- Total active power
- Energy
- Production Ratio
- Power Curve
- Communication status

Alarm triggers

A trigger is the event that sets off the alarm when a site or an element meets a set of activation conditions. Each type of alarm has its own activation conditions. To deactivate the alarm, the element must meet the deactivation conditions defined for that trigger.

> ⓘ NOTE: For more information see the section on Alarm triggers and activation conditions.

Exceptions

The Exceptions feature allows you to retrospectively exclude periods from calculations. This is useful when you need to account for downtimes in production that result from external factors (for example, curtailment requests from the grid operator).

You can apply exceptions at every level of the portfolio, from entire plants, to individual devices. You can also group devices, to account for situations that affect specific areas within a plant.

You can create exceptions in the Budgets panel of the Plant Dashboard module, or from the Alarms table of the Alarms module:

- Create exceptions in the Budgets panel.
- Create exceptions in the Alarms table.

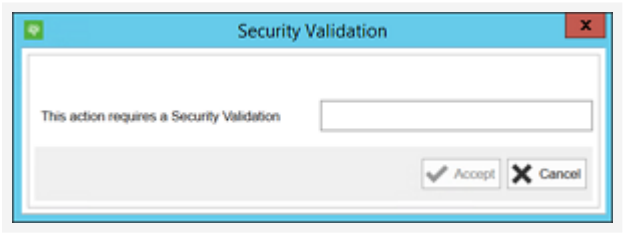
Create exceptions from the Budgets panel

To create exceptions from the Budgets panel of the Plant Dashboard module, follow these steps:

1. On the Budgets panel, click **Exceptions**.

   Result: The **Security Validation** dialog opens:

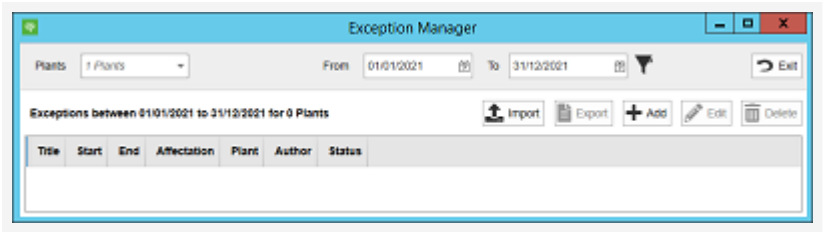
   ### Security Validation dialog

2. On the **Security Validation** dialog, enter the administrator password and click **Accept.**

   Result: The **Exception Manager** dialog opens.

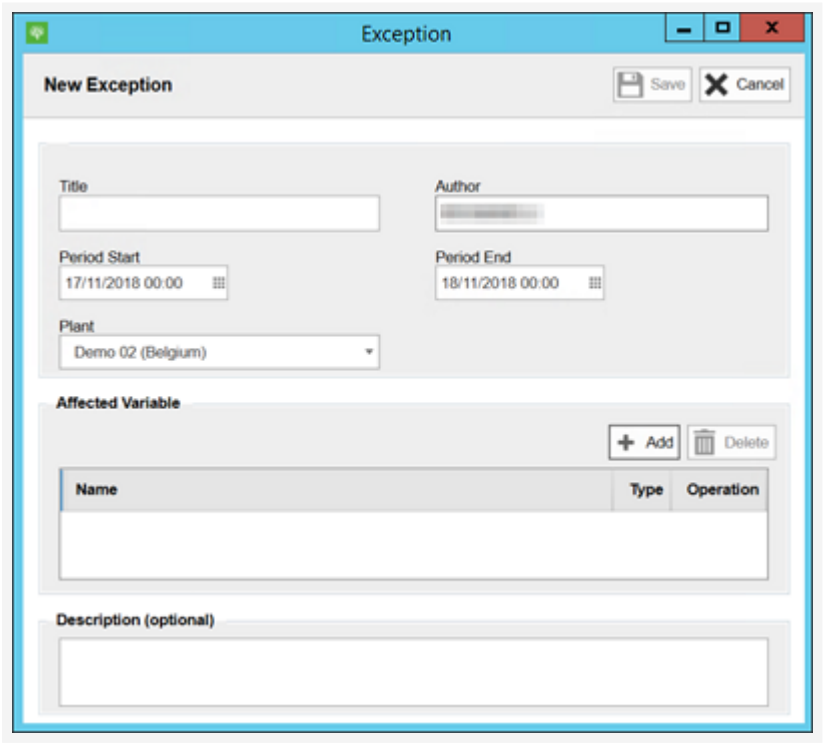
   ### Exception Manager dialog

3. On the **Exception Manager** dialog, click **Add**.

Result: he **New Exception** dialog opens.

New Exception dialog



4. Enter the information for the exception:
   a. *Title*: enter a title for the exception.
   b. Author: displays the username of the person who created the exception.
   c. **Period start**: select the date and time at which the exception starts.
   d. **Period end**: select date and time at which the exception ends.
   e. **Plant**: select the plant to which the exception must be added. This field is pre-filled with the plant from which the exception is being added.
   f. **Affected variable**: click **Add** to select the parameters affected by the exception. For further information, see the instructions to select parameters using the Advanced Datasource Selector.
   g. *Description*: enter a description for the exception.
5. Click **Save**.

Result

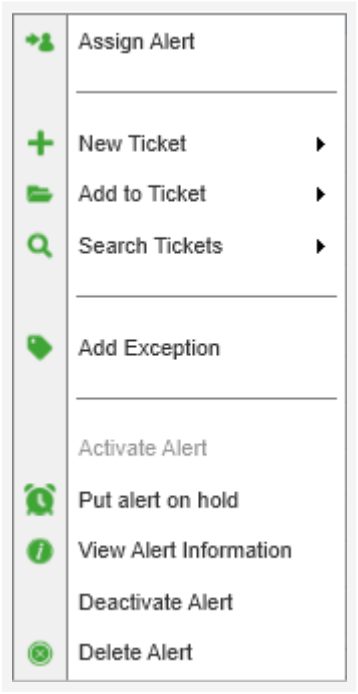
The exception is created.

Create exceptions from the Alarms table

To create exceptions from the Alarms table of the Alarms module, follow these steps:

1. On the Alarms table, right-click the alarm for which you want to create an exception to open the Alarms context menu:

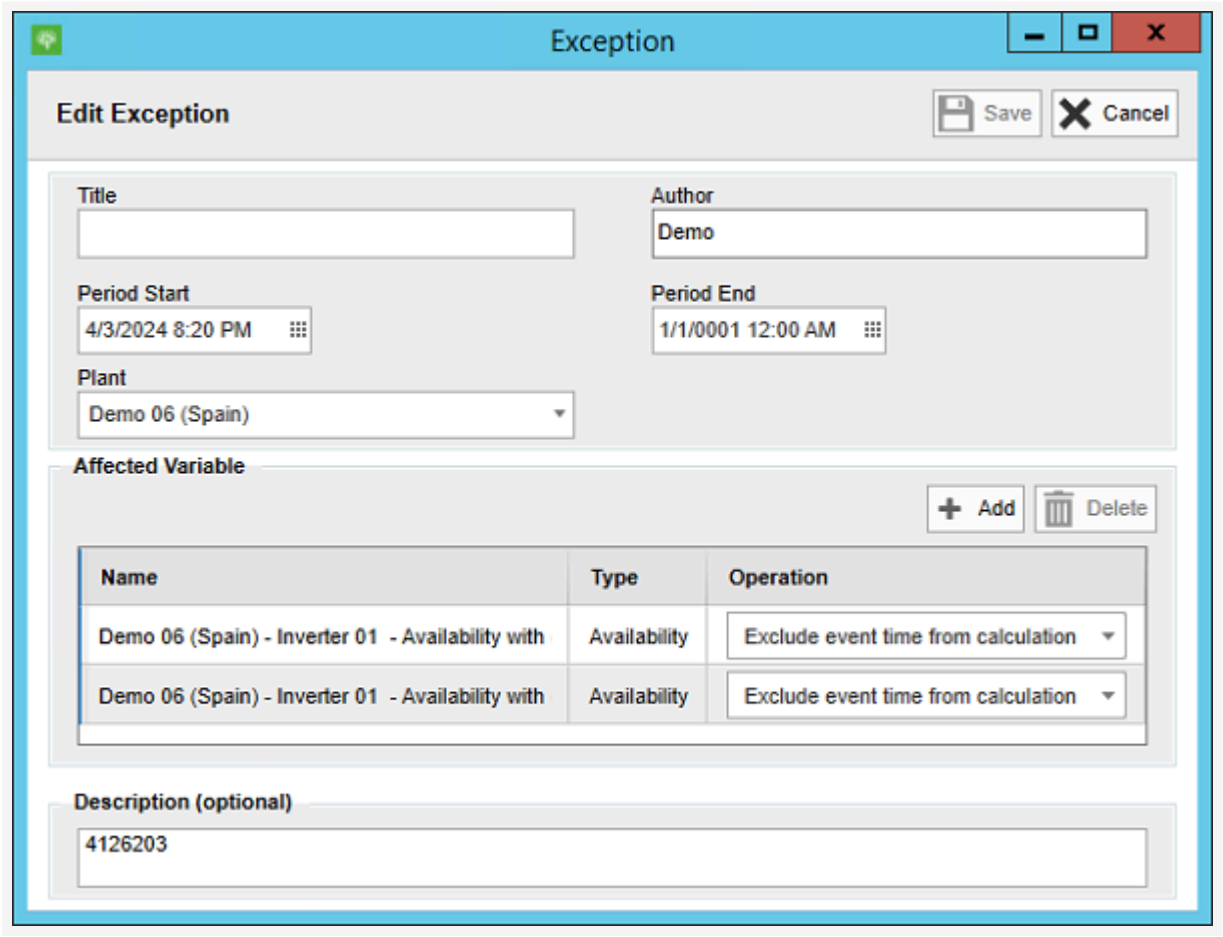
   Alarms context menu

2. Click **Add Exception**.

Result: The **Exception Manager** dialog opens, displaying information for the selected alarm:

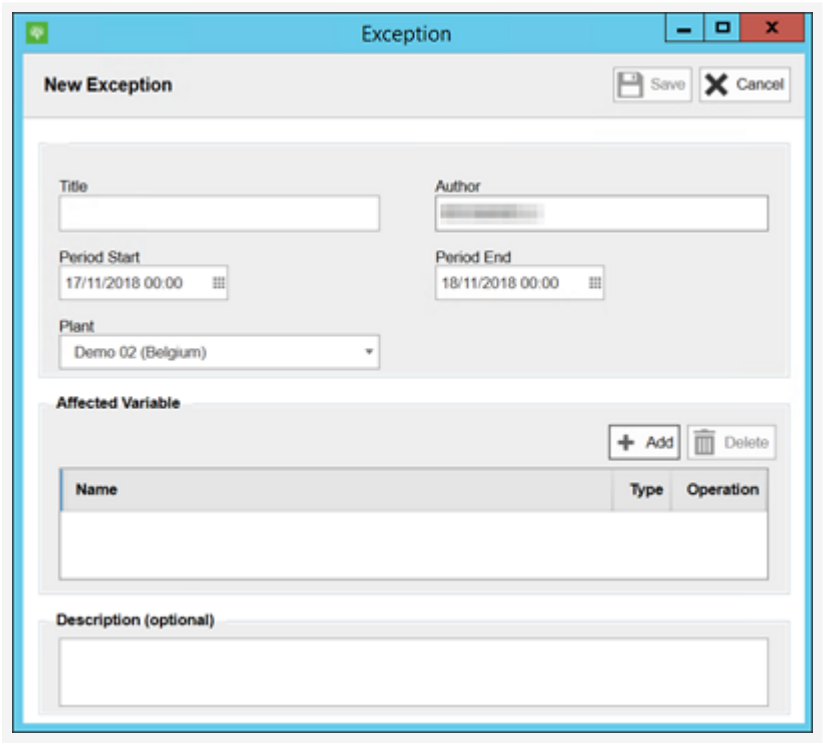
Exception Manager dialog



3 click **Add**.

Result: he **New Exception** dialog opens.

New Exception dialog



4. Enter the information for the exception:
   a. *Title*: enter a title for the exception.
   b. Author: displays the username of the person who created the exception.
   c. **Period start**: select the date and time at which the exception starts.
   d. **Period end**: select date and time at which the exception ends.
   e. **Plant**: select the plant to which the exception must be added. This field is pre-filled with the plant from which the exception is being added.
   f. **Affected variable**: click **Add** to select the parameters affected by the exception. For further information, see the instructions to select parameters using the Advanced Datasource Selector.
   g. *Description*: enter a description for the exception.
5. Click **Save**.

Result

The exception is created.

Modules

GPM SCADA has several modules that you can access from different sections of the user interface.

> ⓘ NOTE: The availability of certain modules, features and functionalities may vary depending on your product configuration.

- Alarms
- Advanced filters
- Commands control
- Data viewer
- Element viewer
- Entity log
- Kiosk view
- Linear Chart viewer
- Live viewer
- Map
- Network Operating Center (NOC)
- Permissions
- Plant
- Plant elements
- Power Plant Control (PPC)
- Ranking
- Reports
- SCADA layout
- Scatter Plot
- Tickets
- Vectorial Layout
- Videowall

Alarms

Alarms are notifications that trigger when certain predefined conditions are met by elements in a plant.

The Alarms module allows you to monitor, manage, and analyze alarms by notifying you when something needs your attention. Alarms can trigger when one or more activation conditions are met. In the Alarms module, triggers are grouped by plant and by alarm.

> ⓘ NOTE: For more information, see the article about Alarm triggers and activation conditions.

The system uses a smart mechanism that constantly checks for new alarms by scanning devices hierarchically, from the highest to the lowest. When the system detects an alarm, it stops checking lower devices to avoid displaying too many Alarms related to the same issue.

> ⓘ NOTE: By default, after an alarm is triggered, the system notifies you every 24 hours that the alarm is still active. It is possible to customize this notification period when configuring the alarm conditions.

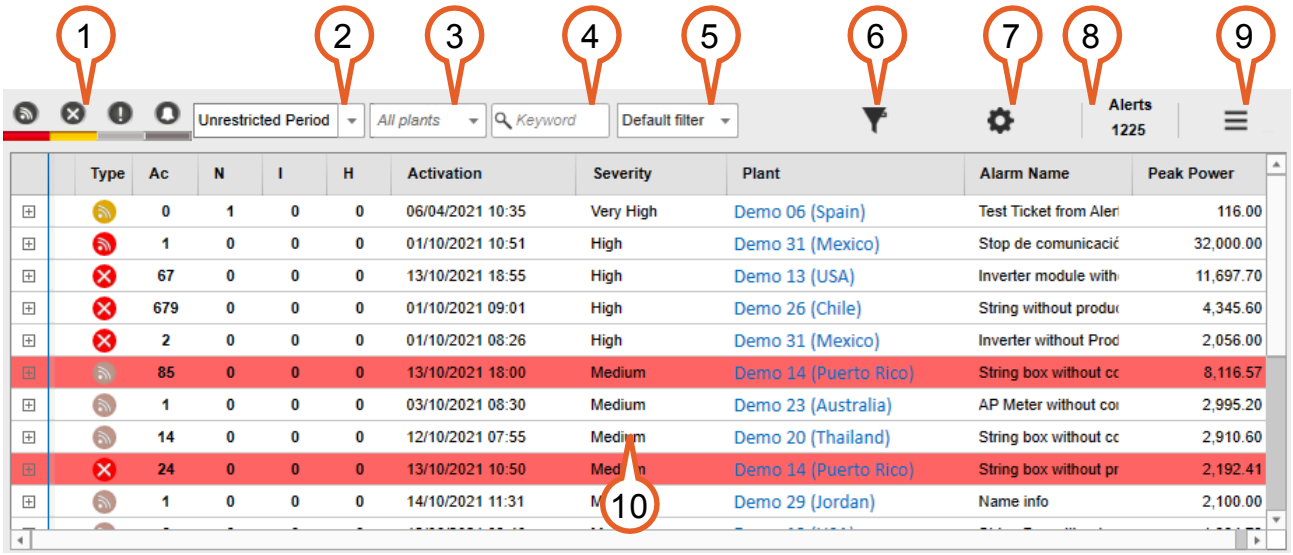
Alarms are also automatically hidden if the issue detected was reported in another alarm.

> ⚠ CAUTION: After you customize your filter, you must click the  to apply the new criteria.

You can open the Context Menu for alarms by selecting one or more alarms and right-clicking them. For more information, see the Alarms Context Menu below.

You can also access the Alarm Information by selecting it from the Context Menu. This menu contains detailed information and data analytics tool for the specific alarm. For more information, see Alarm Information window below.

Alarms module



Type	Ac	N	I	H	Activation	Severity	Plant	Alarm Name	Peak Power
	0	1	0	0	06/04/2021 10:35	Very High	Demo 06 (Spain)	Test Ticket from Aler	116.00
	1	0	0	0	01/10/2021 10:51	High	Demo 31 (Mexico)	Stop de comunicació	32,000.00
	67	0	0	0	13/10/2021 18:55	High	Demo 13 (USA)	Inverter module with	11,697.70
	679	0	0	0	01/10/2021 09:01	High	Demo 26 (Chile)	String without produ	4,345.60
	2	0	0	0	01/10/2021 08:26	High	Demo 31 (Mexico)	Inverter without Prod	2,056.00
	85	0	0	0	13/10/2021 18:00	Medium	Demo 14 (Puerto Rico)	String box without cc	8,116.57
	1	0	0	0	03/10/2021 08:30	Medium	Demo 23 (Australia)	AP Meter without cor	2,995.20
	14	0	0	0	12/10/2021 07:55	Medium	Demo 20 (Thailand)	String box without cc	2,910.60
	24	0	0	0	13/10/2021 10:50	Med n	Demo 14 (Puerto Rico)	String box without pr	2,192.41
	1	0	0	0	14/10/2021 11:31	M	Demo 29 (Jordan)	Name info	2,100.00

1. **Quick filters**: Toggle an alarm type to display or hide related alarm:
   - **Communication**
   - **Stop**
   - **Warning**
   - **Preventive**

   Toggle a status to display or hide related triggers:

Status	Color code	Description
Active		the criteria that caused the alarm activation are still valid and no operator has been assigned to the alarm.
Notified		the criteria that caused the alarm activation are still valid, but an operator has been assigned to the alarm.
InactiveInactive		the alarm deactivation criteria have been met.
On hold		a user put the alarm on hold for a specific period of

time.

ⓘ NOTE: The default icons and colors are customizable. Contact your GPM representative if you want to change them.

2. **Period picker**: select a date range from the drop-down list to display alarms that were activated during that period:
   - **Current day**
   - **Custom period**
   - **Last 15 days**
   - **Last 30 days**
   - **Last 7 days**
   - **Unrestricted period**

   ⓘ NOTE: When you choose **Unrestricted Period**, the system automatically disables active alarms to avoid generating too many queries.
3. **Plant filter**: select a plant from the drop-down menu to display only the alarms related to it.
4. *Keyword filter*: enter a keyword to filter alarms by their name, Alarm ID, or global Alarm ID:
   - Use # before a number to search by Alarm ID.
   - Use * before a number to search by global Alarm ID.
   - Use ; to separate multiple criteria.
5. **Saved filters**: select a filter from the drop-down menu to apply it to the alarm list.
6. **Apply filters**: click to apply the combination of filters you have selected.
7. **Advanced filters**: click to display the Advanced Alarms Filters window and customize your filter with more advanced options. For further information, see Advanced Alarm Filters.
8. Total tiggers: displays count of the alarms that are currently displayed.
9. **More actions**: click to display more actions:
   - **Export Alarms**: click to export alarms that are currently displayed to a Microsoft Excel format. For further information, see Export Data to File.
   - **Audible Alarms**: toggle to enable or disable audio alarms.

- **Blinking Alarms**: toggle to enable or disable blinking alarms.
- **Group Alarms**: toggle to group or ungroup the alarms. When alarms are ungrouped, you can select multiple triggers and perform bulk actions on them.

10. Alarms table: displays groups of alarms with their triggers, automatically collapsed, and sorted in descending order by severity, status and activation time.

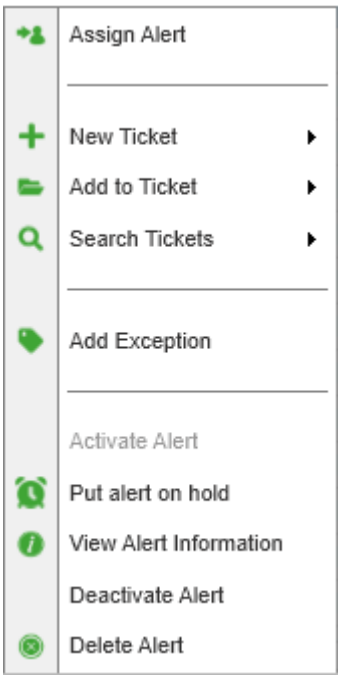
    Double-click a group of alarms to expand it and display related triggers.

    Click a plant to open its hierarchy tree in the Navigation module.

    Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers.

    Right-click a trigger to open the context menu and perform quick changes.

Context menu



Option	Description
Assign alarm	Click to open a dialog that allows you to assign the alarm to an operator.
New ticket	Select a new ticket type to create a new ticket and automatically pair the selected element to it.
Add to ticket	Place your cursor here, enter a ticket ID in the search field, and press enter to add the selected element to an existing ticket.
Search tickets	Choose a ticket type to open the related tickets in the Tickets view.
Add exception	Click to create an exception and add the alarm ID in the exception description. For further information, see the Commands section.
Put alarm on hold	Click to put the alarm on hold. You must specify the time period for which the alarm remains on hold. Once the time is up, the alarm reverts to its previous status. You can also input the reason for putting the alarm on hold.

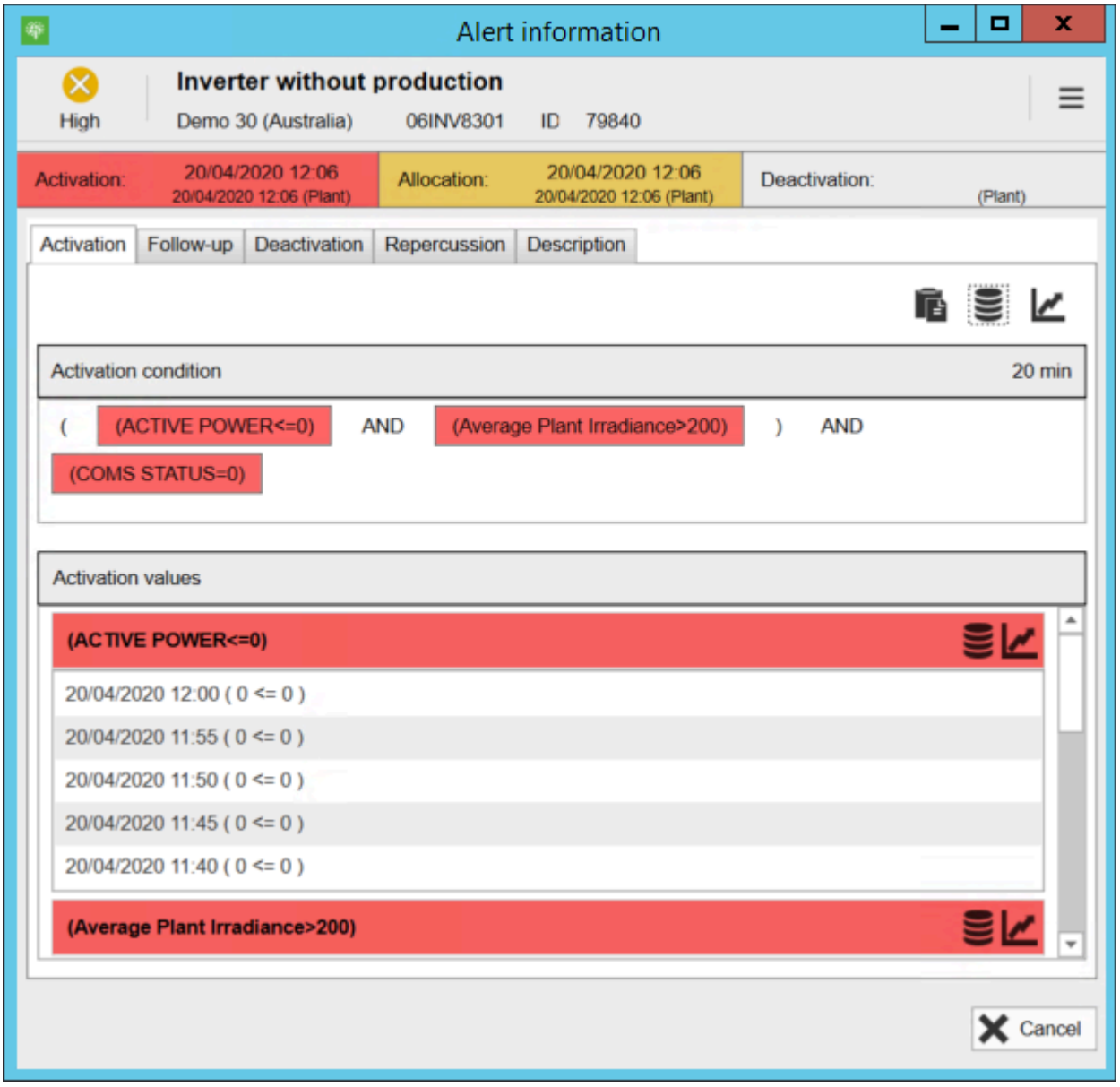
	It is also possible to delete an existing hold. This resets the alarm to its previous status.
View alarm information	Click to open the Alarm Information window and see further information about the alarm.
Deactivate alarm	Click to deactivate the alarm. ⚠ CAUTION: This action requires the administrator password.
Delete alarm	Click to delete the alarm. ⚠ CAUTION: This action requires the administrator password.

Alarm Information window

In the Alarm Information window, the upper displays the alarm priority and the timestamp of Activation, Allocation, and Deactivation, which are the core events of an alarm. In the upper area, you can also create tickets and automatically link them to the alarm by clicking the ☰ icon.

The area below the upper bar is organized in tabs. Each tab contains a different piece of information. When it is possible to perform actions on the data displayed, the tab will include Action buttons.

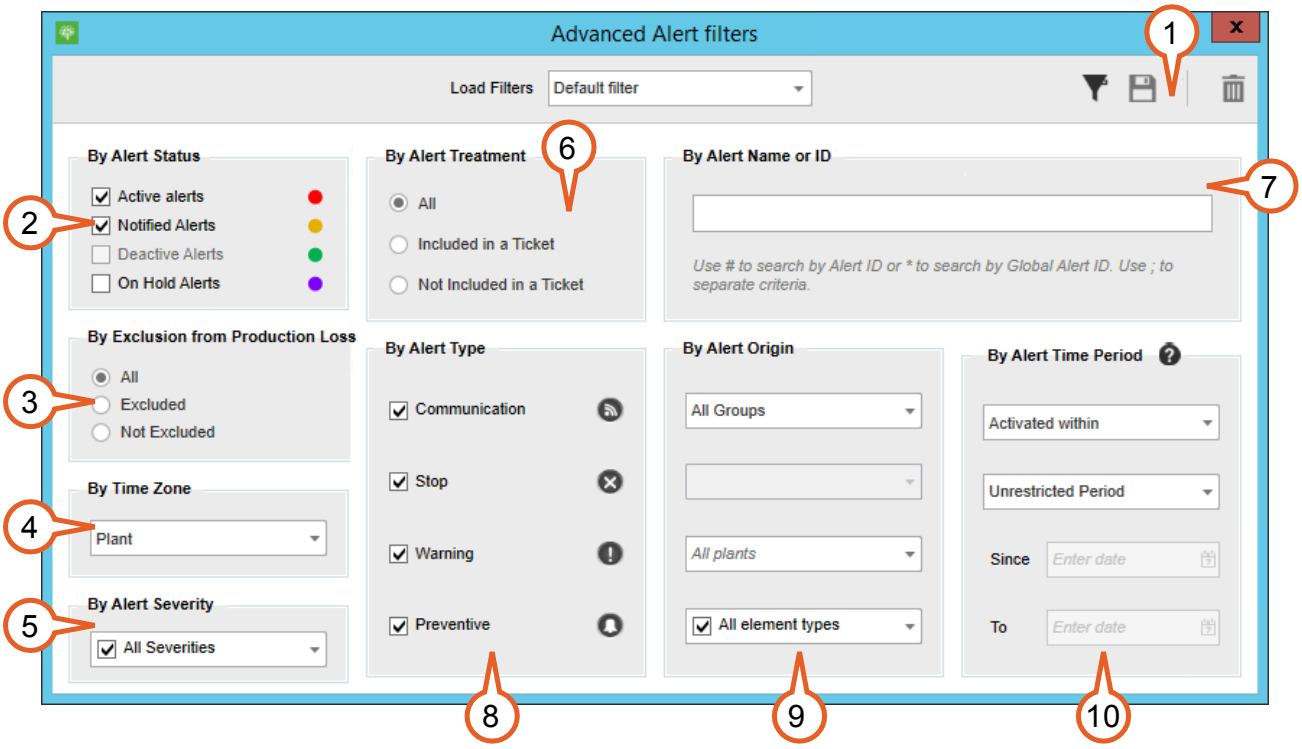
Alarm Information



Tab	Description

Activation	Includes the activation condition and the latest values of each criteria used in the activation condition.
Follow-up	Includes all the actions performed by operators on the trigger since it was triggered.
Deactivation	Includes the deactivation condition and the most recent values of each criteria used in the deactivation condition.
Repercussion	When the triggering of the alarm affects availability, users can see the timestamp of the issue as well as its total duration. If the status of the trigger is "On hold", this tab also includes the time that the alarm has been on hold.
Description	Includes the alarm description.

Advanced alarm filters



1. Action buttons: click to perform actions on the alarm:
   - **Apply**: apply the filters to the list without saving.
   - **Save**: save the filters for future use.
   - **Delete**: delete the selected filter.
2. Status: select the alarm status:
3. Production loss exclusion: select whether to include or exclude alarms that caused losses in production.
4. Time zone: select the time zone for the plant.
5. Severity: select the alarm severity. You can select more than one option.
6. Treatment: select whether the alarm have been included in a ticket.
7. Name or ID: enter the alarm name or ID.
8. Type: select one or more alarm types.
9. Origin: select the plants or devices that the alarm is associated to.

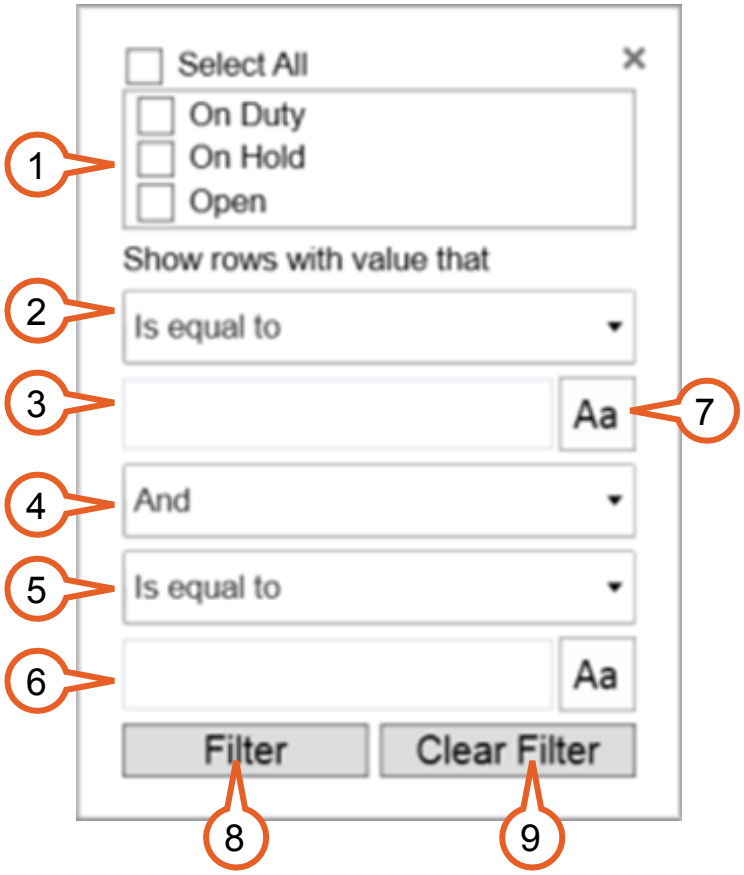
ⓘ NOTE: The alarm origin allows you to filter alarms by plants or devices using the pre-defined groups of your portfolio.

10. Period: select the alarm time criteria and period.

Advanced filters

Advanced filters allow you to filter the content of specific columns in a table using two mechanisms. The first mechanism allows you to select the values that you want to display from a list containing all the values of a column. The second mechanism allows you to apply a simple Boolean expression with two conditions to filter the values of a column.

Advanced filter



1. **Available values**
2. **First Boolean condition**
3. *First values*
4. **Boolean operator**
5. **Second Boolean condition**
6. *Second values*
7. **Case-sensitive button**
8. **Apply filter**
9. **Clear filtering criteria**

Use advanced filters

To use advanced filters, follow these steps:

1. Click the icon on any column where the advanced filtering option is available.
   Result: The **Advanced Filtering** dialog appears.
2. (Optional) Select the values that you want to display on the table.

   > ⓘ NOTE: Selecting values automatically displays them on the table.

3. In the **Show rows with value that** section, select the first Boolean condition from the drop-down list.
4. In the first *Values* input field, enter the first values.

   (Optional) Toggle the Aa button to turn the case-sensitive option on or off.

   > ⓘ NOTE: The case-sensitive button is not available when you are filtering numerical values.

5. Click the second **Boolean operator** drop-down menu and select the second Boolean condition from the drop-down list.
6. In the second *Values* input field, enter the second values.

   (Optional) Toggle the Aa button to turn the case-sensitive option on or off.

   > ⓘ NOTE: The case-sensitive button is not available when you are filtering numerical values.

7. Click **Filter**.

Result

The filter is applied and elements matching the filtering criteria are displayed on the table. The icon on the column turns blue.

> ⓘ NOTE: You can clear all the criteria by clicking **Clear Filter**.

Commands Control

The Commands Control module allows you to monitor, manage, and send pre-configured command sequences to the physical elements in your portfolio. The module is divided in three tabs:

- Commands
- Command History
- Command Retry Queue

⚠ CAUTION: Follow all the safety procedures before performing any action from this module. These actions directly impact the plant.

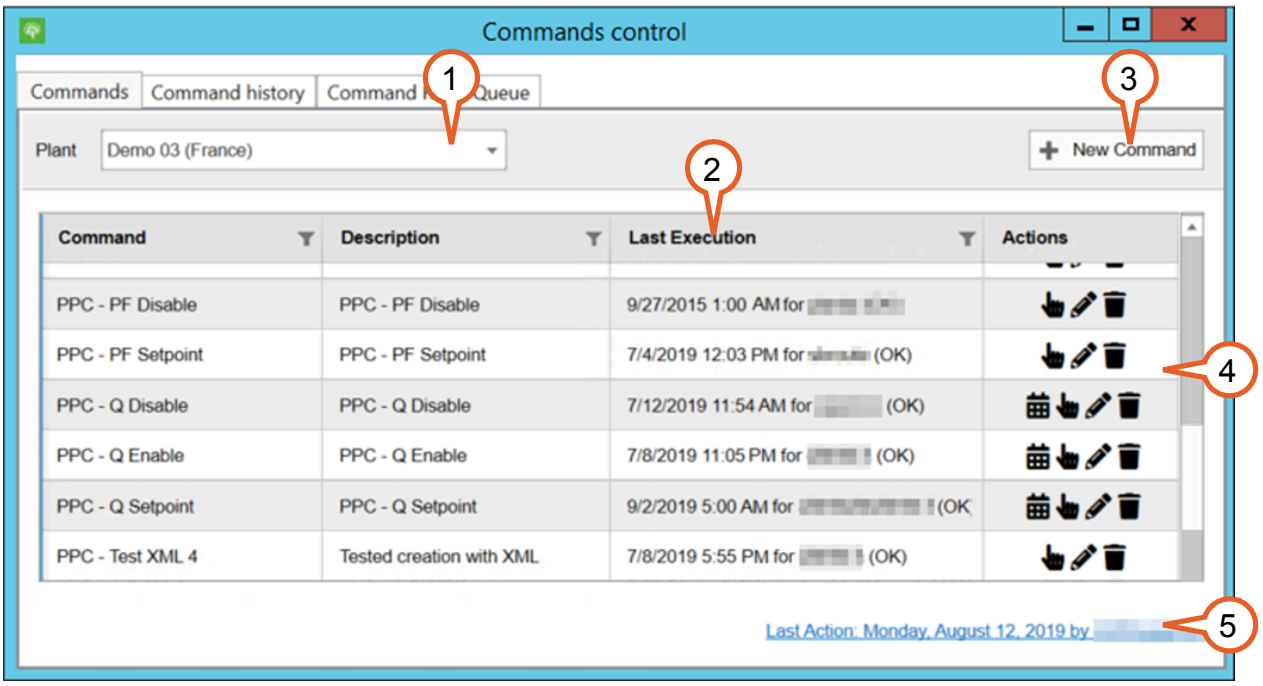
⚠ CAUTION: For security reasons, actions taken in the Commands Control module are protected by a password and stored in the application log.

To access the Commands Control module, click the 👆 icon on the Upper Bar.

Commands

The Commands tab allows you to create, send, edit, and delete command sequences.

Commands tab



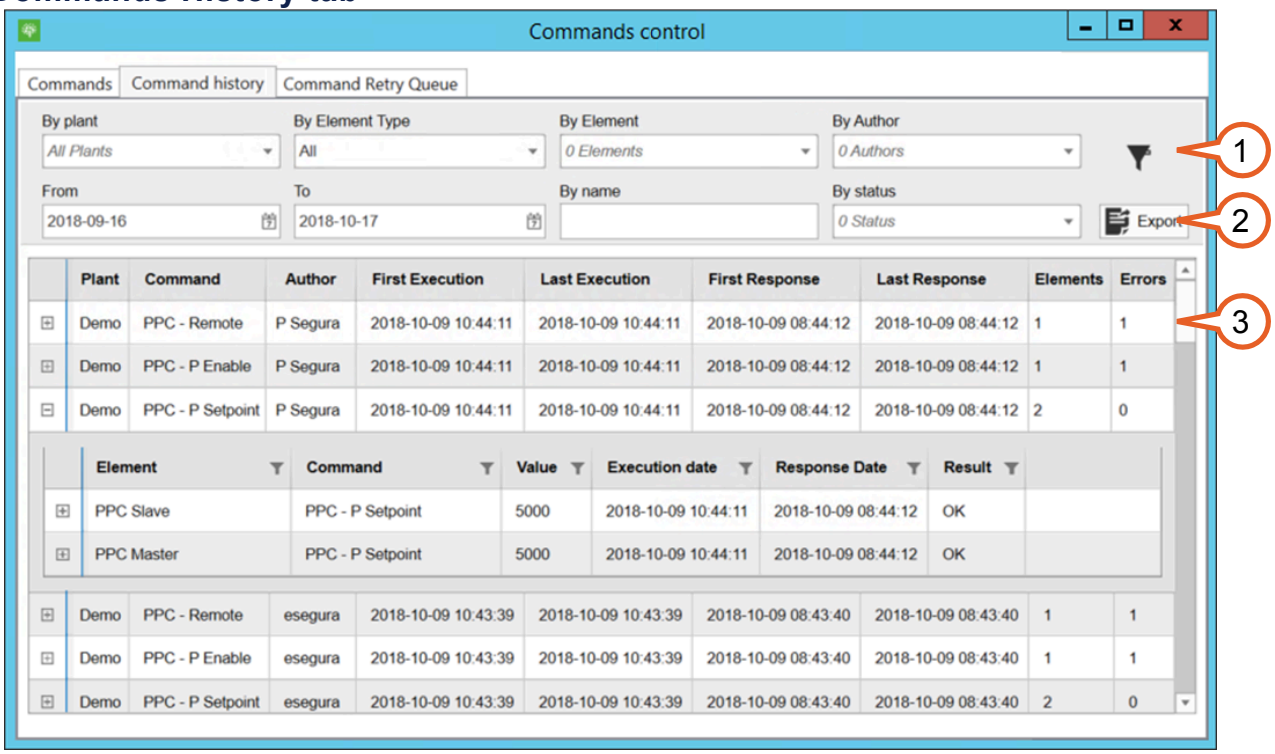
1. **Plant selection**: click to select a plant from the drop-down list to display the available commands.
2. Commands list: displays the available commands for the selected plant.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon ▼ on any column header for advanced filtering. For further information, see Advanced Filters.
3. **New command**: click to create a new command sequence. For further information, contact your GPM representative.
4. **Action buttons**:
   - **Execute** the command.
   - **Edit** the command sequence.
   - **Delete** the command.
   - **View or edit** the scheduled command.
5. **Last action**: displays the date and the user who made the last changes to the Command Controls view.

> ⓘ NOTE:
>
> Clicking on the hyperlink takes you to the Actions Log.

Commands History

The Command History tab allows you to see the history of the commands that have been executed and export this information.

Commands History tab



1. **Filter**: select the filtering criteria and click ▼ to display commands that match the criteria on the list.
2. **Export**: click to export the list to a Microsoft Excel format. For further information, see Export Data to File.
3. Commands list: click a column header to sort the table by the values of that column. You can rearrange columns by dragging and dropping the headers.

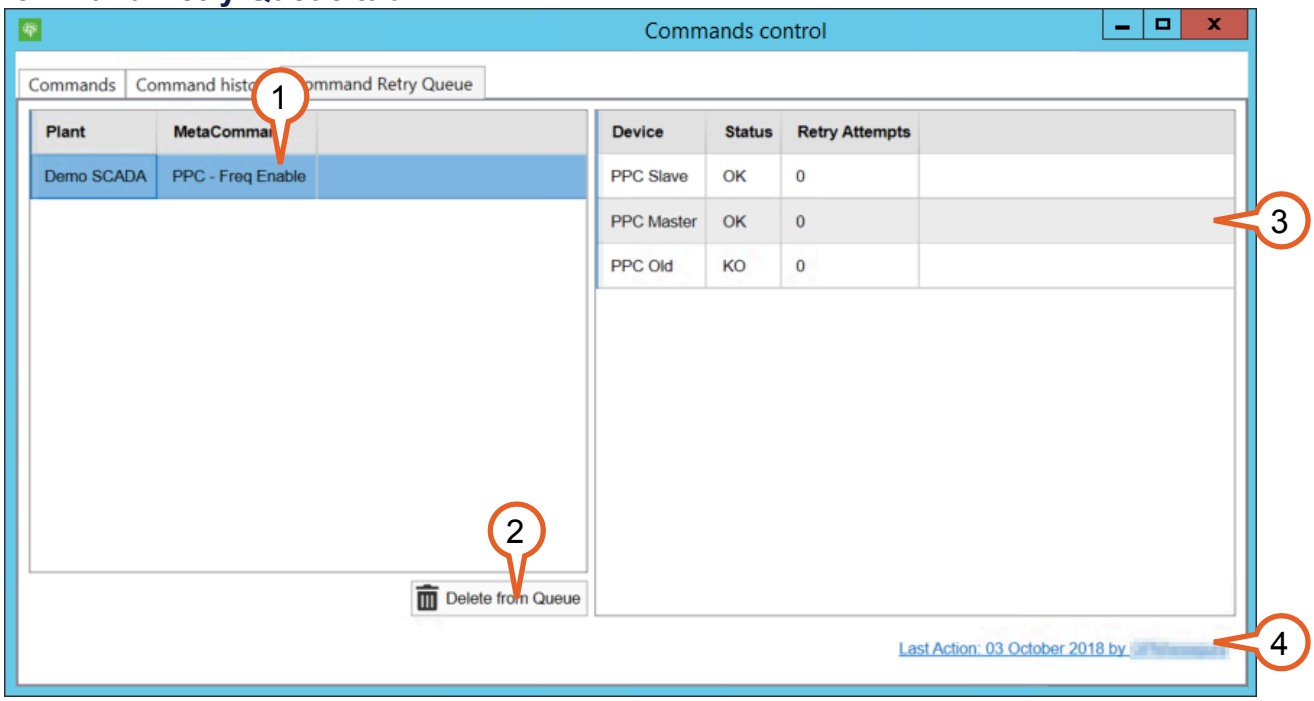
Command Retry Queue

The system can be configured to resend a command when it is not possible to communicate with the device.

The Command Retry Queue tab displays the commands that are queued for a retry when they fail to be sent. If you do not want the system to retry sending a command, you can delete it from the retry queue.

> ⓘ NOTE: The Command Retry Queue tab is only available for plants with multiple PPCs.

Command Retry Queue tab



1. Retry queue: displays commands that are being retried are displayed here. Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers.
2. **Delete**: click to delete the selected commands from the retry queue.

   > ⓘ NOTE: This action requires the administrator password.

3. Command sequence: displays the retry status of each PPC to which the command is sent.

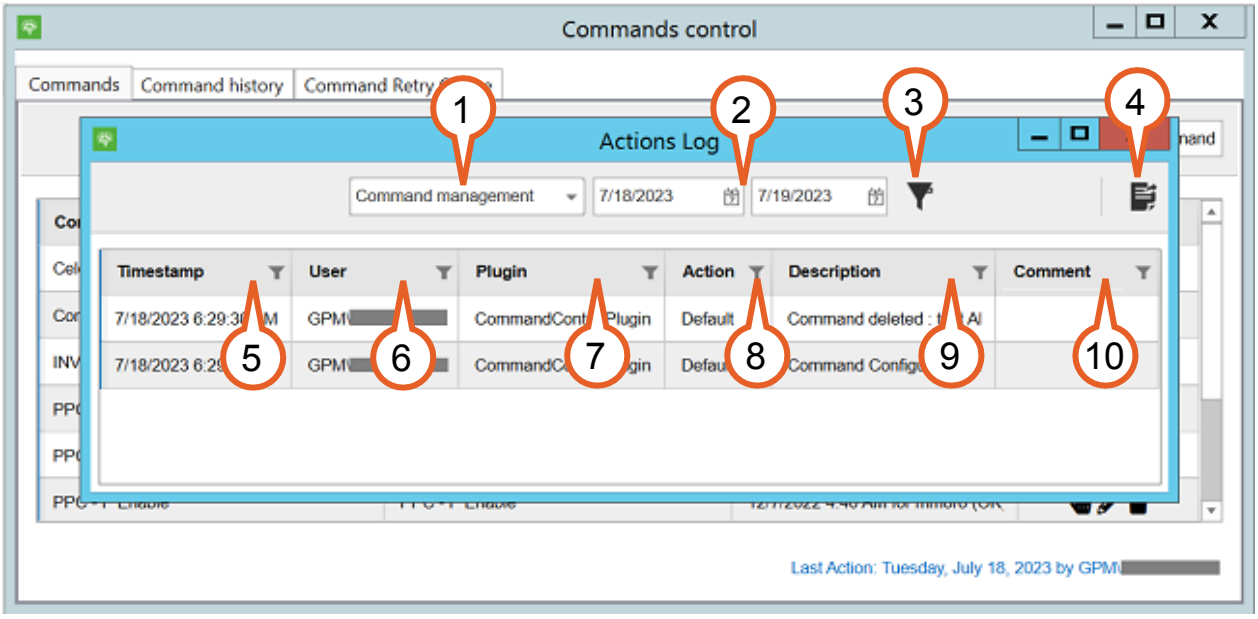
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers
4. **Last action**: displays the date and the user who made the last changes to the Command Controls module.

> ⓘ NOTE:
>
> Clicking on the hyperlink takes you to the Actions Log.

Actions Log

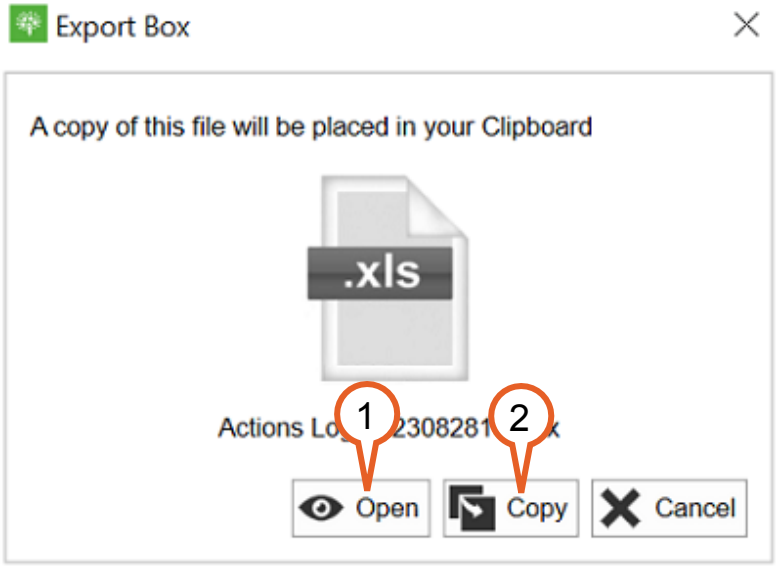
The Actions log records all users activities in the system. It includes action timestamps, user identification, action descriptions and classification by modules.

Actions log



1. Module: filter the actions by module.
2. Calendar:filter by the dates selected in the calendar.
3. Filter: apply the module and calendar filters.
4. Export: access the Export Box to view or download the Actions log.
5. Timestamp: date and time when the action was performed.
6. User: user that performed the action.
7. Plugin: plugin related to the action.
8. Action: type of action.
9. Description: brief description of the action.
10. Comment: comments related to the action.

Actions log export



1. Open: see the Actions Log in XLS format.
2. Copy: download an XLS copy of the Actions log onto your Clipboard.

Data viewer module

The Data Viewer is a tool that allows you to create queries and analyze the data of your portfolio. The results of the query are arranged in tables that display the values for the selected parameters at specific moments in time. You can save the results of your queries for further use or export them to other views.

You can add queries to the Data Viewer from other areas of the user interface or create queries directly from the Data Viewer module.

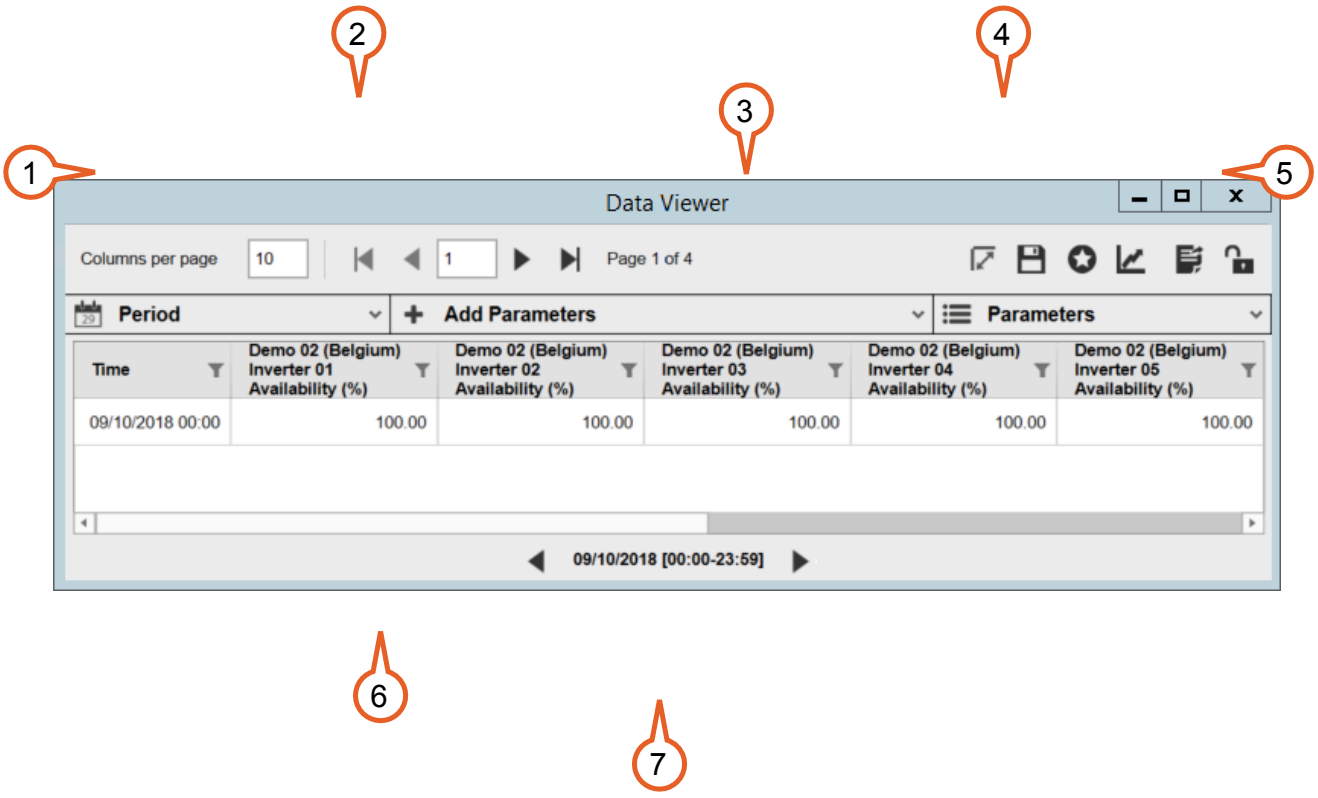
Access the data viewer by clicking the icon on the upper bar of the user interface.

> ⓘ NOTE: You can open multiple Data Viewer module windows, but you can only add queries to an unlocked window. The active window has the icon at the top right corner.

The Data Viewer module has three panels to define the information on display:

- Query period
- Add parameters
- Current parameters

Data Viewer module

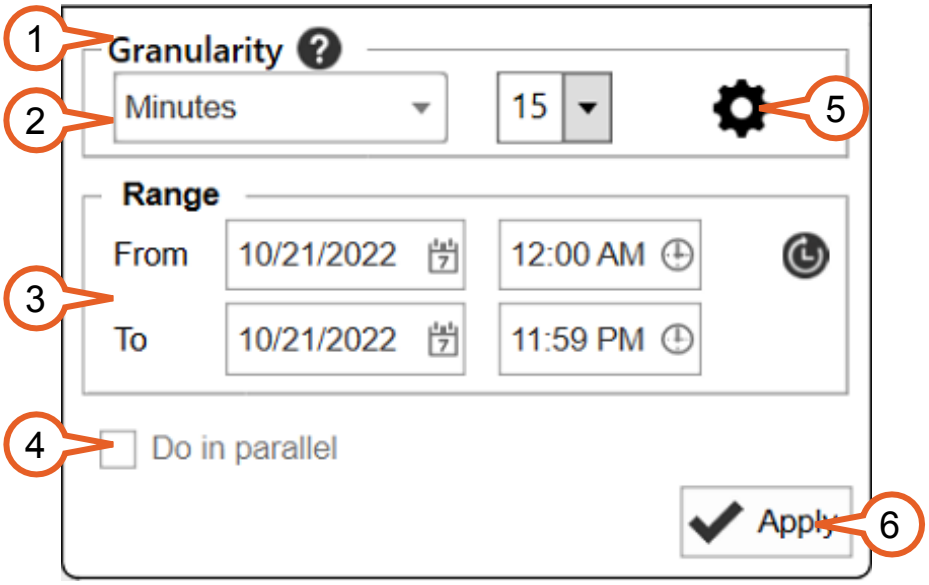


1. Query period: click the icon to expand the menu and customize the data granularity and time range. For more information, see Query period.
2. Pagination: navigate between pages using the arrows and the text input field and define the number of columns to display on each page.
3. Add parameters: click the icon to expand the menu and add parameters to the table. For more information, see Add parameters.
4. Action buttons:
   - Invert the axis of the table.
   - Save the current query to your favorites.
   - Load a favorite query.
   - Analyze the selected parameters in the Linear Chart Viewer module.
   - Export data to the Live viewer module.
   - Export the query to the clipboard or to a file.
   - Toggle the window lock on and off. When you lock a window, you cannot add further parameters to it from other parts of the application.
5. Current parameters: click the icon to expand the menu and manage the parameters currently on display. For more information, see Current parameters.
6. Data table: click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers.
   Click the icon on any column header for advanced filtering. For further information on advanced filtering, see Advanced Filters.
   When you are analyzing consolidated data (for example, datasets of values aggregated every 5 minutes), you can right-click one or more selected values and click the icon to display the data in the Live Viewer module. For more information about data consolidation intervals, see Data Consolidation.
7. Period browser: browse the period using the arrows. The period displayed is based on the query period that you selected.

Query Period

The Query Period panel lets you customize the data granularity and time range of your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Query Period panel



1. **Data granularity**: leave the default data granularity setting or select a data granularity from the drop-down list.

   For example, to see data in 15-minute increments, select "Minutes" and "15" in the drop-down menus.

2. **Force recalculation**: click to manually force a recalculation of the data if the values have changed you do not want to wait for the next automatic recalculation to see them. The system automatically calculates values at recurring intervals and stores them in the database for faster access.

   This option is only available when the selected granularity is **Days**, **Months**, or **Years**.

   ⓘ NOTE: Recalculations may take several minutes.

3. **Date and time range**: select the date and time range using the calendar picker. Click the ⏱ icon to set the range to the default period.
4. **Do in parallel**: select the checkbox to analyze the same parameter over equivalent periods. Each period is displayed as a column.

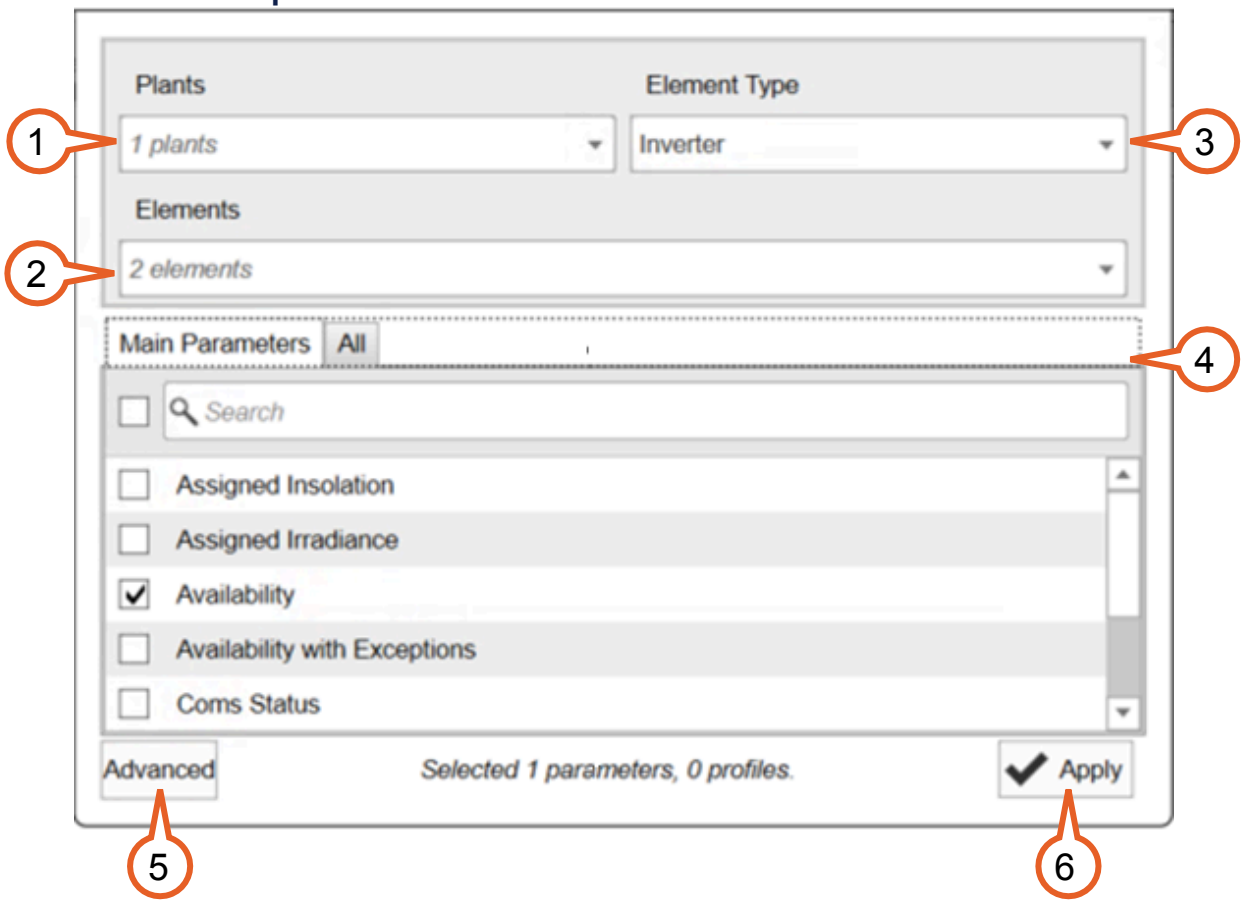
ⓘ NOTE: This option is only available when the data granularity unit is smaller than the period unit.

5. **Data granularity settings**: Click to access advanced settings for data granularity, such as grouping methods to aggregate data, and the operations you want to perform on it (for example, calculating averages).
6. **Apply**: click to apply your settings to the query.

Add Parameters

The Add Parameters panel lest you add parameters to your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Add Parameters panel



1. **Plants**: click to select one or more plants from the drop-down list. You can use the *Search* field to refine the drop-down list results.
2. **Elements**: click to select one or more element from the drop-down list.

   ⓘ NOTE: You can only select elements after selecting multiple plants.

3. **Element type**: click to select one or more element type from the drop-down list.

   ⓘ NOTE: You can only select element types after selecting multiple plants.

4. Parameters: click to select one or more parameters or series from one of the tabs. You can use the *Search* field to refine the drop-down list results.
   - **Main Parameters**: lists the most relevant parameters that are configured for

the selected element.

- **All**: lists all the parameters that are retrieved from the selected element.

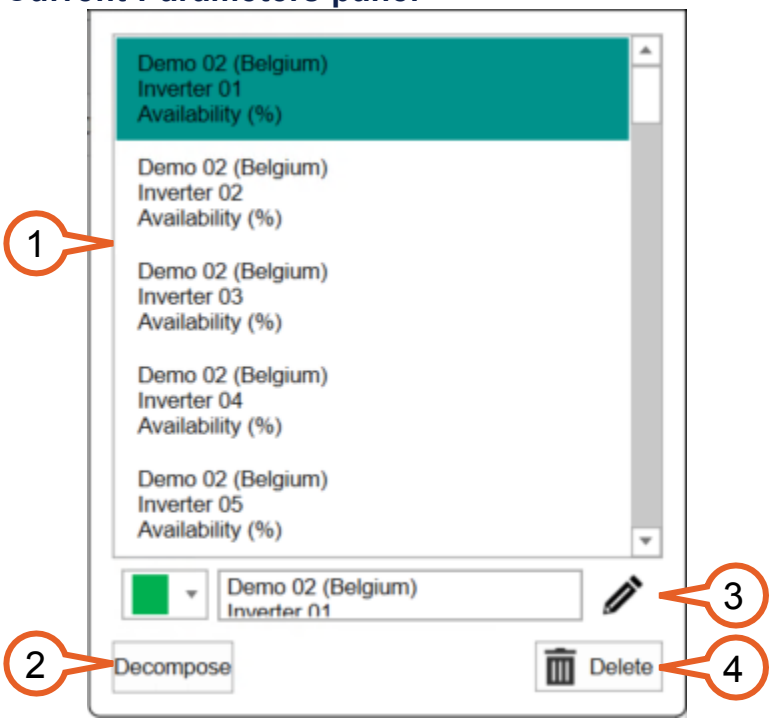
5. Advanced mode: Click to open the Advanced Datasource Selector window. For further information, see the Advanced Datasource Selector.
6. **Apply**: click to apply your settings to the query.

Current Parameters

The Current Parameters panel lets you the manage parameters included in a query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons. You can also change the display name and the color of a parameter.

It is possible to see a break-down of parameters to analyze complex data in detail.

Current Parameters panel



1. Parameters: lists the parameters currently displayed on the table. Select one or more parameters to customize their appearance. To select multiple parameters, use CTRL+Click.
2. **Decompose**: click to add all the factors that make up the selected parameter to the query. Factors can be other parameters or data sources.
3. **Edit**: click to enable editing for the selected parameter. You can customize the background color and the column heading text. The default text has the plant name, the element name, and the parameter name.
4. **Delete**: click to delete the selected parameters from the chart.

Element viewer module

The Element Viewer module is a dynamic tab that allows you to quickly monitor, analyze, and perform maintenance on a selected plant or device.

> ⓘ NOTE: The information displayed on the Element Viewer refreshes automatically when GPM SCADA receives data from the selected element.

You can perform tasks to manage elements directly from this module.

Element Viewer module

1. Heading: When you select a:
   - **Plant**: displays the plant name, coordinates and peak power.
   - **Device**: displays the device name, the plant it belongs to, and its peak power. If there is an active alarm on the selected element, you can click the alarm to open the Alarm Information window.
2. **Tabs**: select a tab to display its content on the Main Area. For further information,

see the following sections:

- Monitor
- Live values
- Parameters
- Info
- Commands

3. **Quick actions**: take quick actions related to the selected plant or device.For further information, see Quick Actions below.
   - Ticket menu
   - View Element menu
   - Copy to Clipboard
   - Refresh
   - Copy Window
4. Main area: displays information about a selected element.

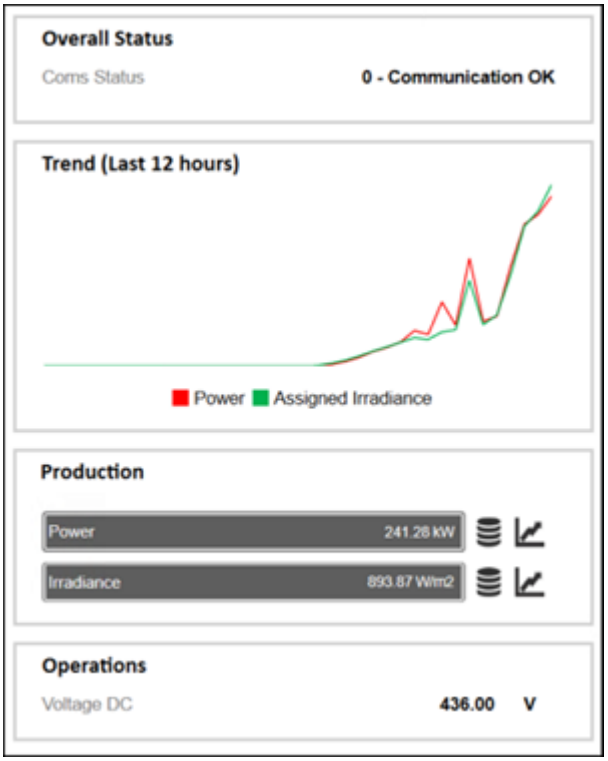
Monitor tab

The Monitor tab provides you with tools to monitor the main key performance indicators (KPIs) of a selected element. The Monitor tab tools are organized in cards and change based on the selected element type.

Access the Monitor tab by clicking the icon on the Element Viewer.

You can place your cursor on any chart heading and click the icon to display the chart KPIs in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.

Monitor tab



Live Values tab

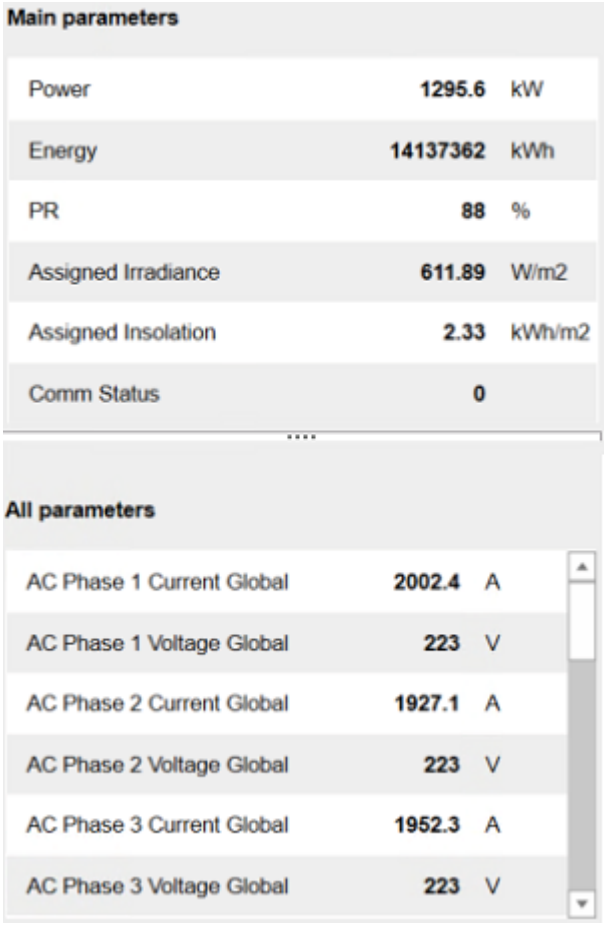
The Live Values tab allows you to monitor data that is retrieved in real time from a selected element.

> ⓘ NOTE: Real-time data availability depends on the system capability of communicating data in real time.

Access the Live Values tab by clicking the icon on the Element Viewer.

You can place your cursor on any parameter with real-time data and click the icon to display its values in the Live Viewer module. For further information, see Live Viewer Module.

Live Values tab



> ⓘ NOTE: You can resize the **Main parameters** and **All parameters** sections by clicking and dragging the dotted line that separates them. The system saves your preferences after the change.

Section	Description
Main parameters	Most relevant parameters that are configured for the selected element. ⓘ NOTE: If you want to modify the parameters displayed here, contact your GPM representative.
All parameters	All the parameters that are retrieved from the selected element.

Parameters tab

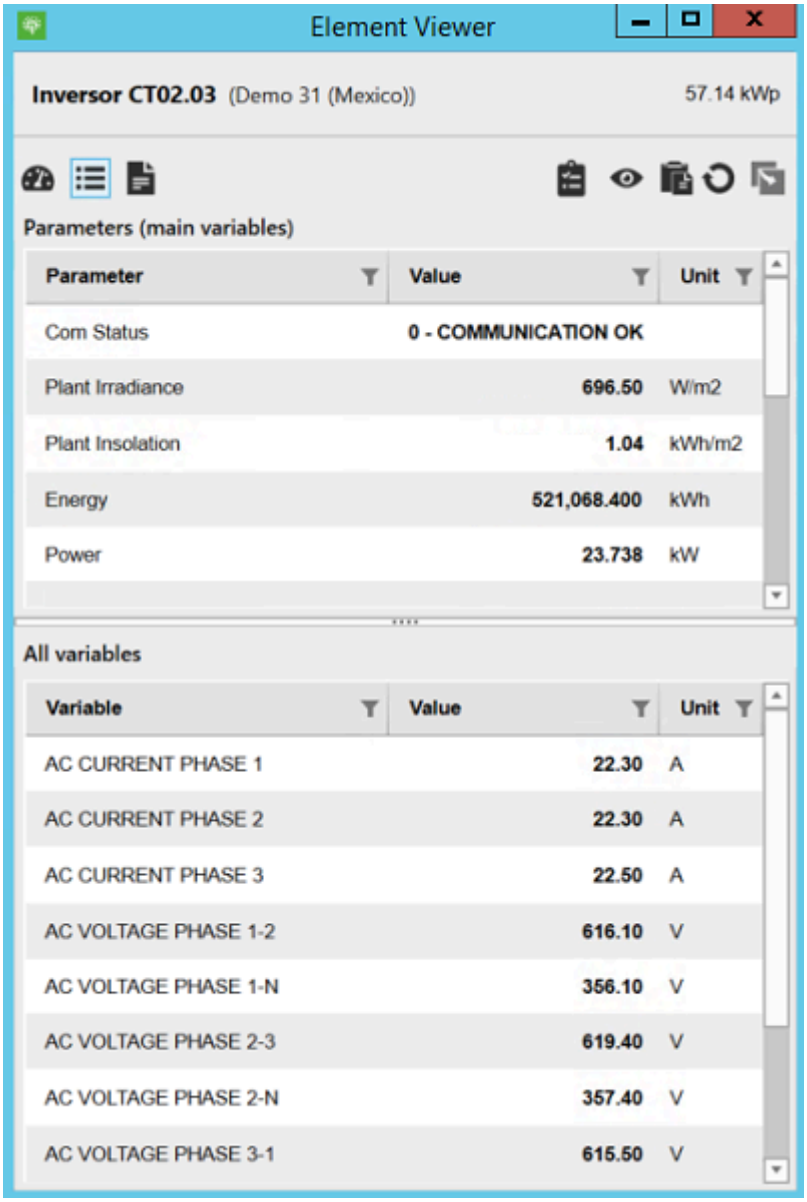
The Parameters (main variables) tab allows you to monitor data that is retrieved at regular time intervals from the selected element.

Access the Parameters tab by clicking the icon on the Element Viewer.

You can place your cursor on any parameter and click the icon to display the parameter values in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.

You can click the icon on the column headers to open the filtering options for the data on display. This allows you to find, visualize, and select specific variables. For more information about filtering options, see Advanced Filtering.

Parameters tab



ⓘ NOTE: You can resize the **Parameters (main variables)** and **Variable** sections by clicking and dragging the dotted line that separates them. The system saves your preferences after the change.

Section	Description
Main Parameters	Most relevant parameters that are configured for the selected element.

ⓘ NOTE: If you want to modify the parameters displayed here, contact your GPM representative.

All Variables	All the variables that are retrieved from the selected element. ⓘ NOTE: This section is only available for elements.

Info tab

The Info tab provides contextual information about the selected plant or device. The information is retrieved from the entity's metadata, which is customizable and can change depending on your setup.

Access the Info tab by clicking the element icon in the Navigation module or by clicking the  icon on the Element Viewer.

You can right-click a parameter and select **See History** to open the Entity Log, where you can track and manage changes made the values. For further information, see Entity Log.

Info tab

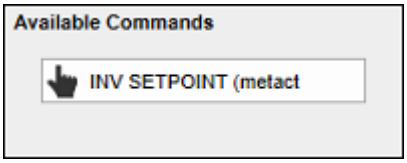
Description Parameters	
Name	INV 15B
Typology	Inverter
Description	00000DCE/000021FC/2

Commands tab

The Commands tab allows you to send pre-configured commands to one or more devices. When you click a command on the list, the **Command Execution** dialog opens.

Access the Commands tab by clicking the icon on the Element Viewer.

Commands tab



Quick Actions

The quick actions buttons allow you to easily perform basic tasks directly from the Element Viewer. Click any icon to access the available options:

- **Ticket menu**: pair the selected element to a ticket or find existing tickets related to the element.

Option	Description
New Ticket	Create a new ticket and automatically pair the selected element to it: ▪ Create maintenance tickets ▪ Create task tickets ▪ Create data correction tickets
Add to Ticket	Choose a ticket type to open the related tickets in the Tickets module.
Search Tickets	Choose a ticket type to open the related tickets in the Tickets module.

- **View Element menu**: display information related to the selected element.

> ⓘ NOTE: The options available when you click the View Element button depend on the selected element.

Option	Description
Alarm Grid	Display related alarms on the Alarms tab of the Content Area.
Search Tickets	Choose a ticket type to open the related tickets in the Tickets module. ⓘ NOTE: This option is only available when you are viewing a plant in the Element Viewer.
Portfolio Map	Display the plant on the Portfolio Map tab of the Content Area.

ⓘ NOTE: This option is only available when you are viewing a plant in the Element Viewer.

Commands Log	Display related commands in the Commands Control module.

- **Copy to clipboard**: click to copy all the parameters and values on display to your clipboard.
- **Refresh**: click to refresh te information displayed in the tab.
- **Copy Window**: click to open a copy of the Element Viewer module in a separate window.

Element management

You can manage elements directly from the Element Viewer.

Tickets

- Create maintenance tickets
- Create task tickets
- Create data correction tickets
- Add elements to existing tickets

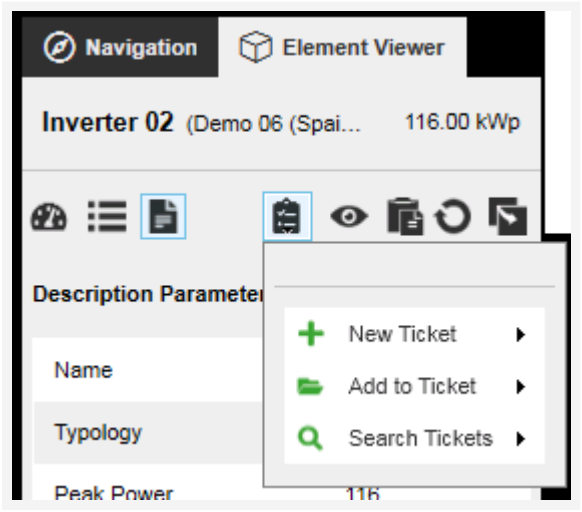
Create maintenance tickets from elements

To create a maintenance ticket from an element, follow these steps:

> ⓘ NOTE: Information contained in Maintenance tickets is customizable. Each customization is stored as a template. The following instructions use the **GPM Default** template.

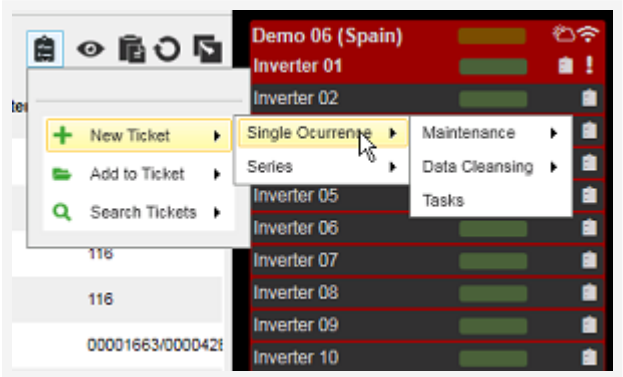
1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

Ticket menu



2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.
   Result: The ticket type panel opens:

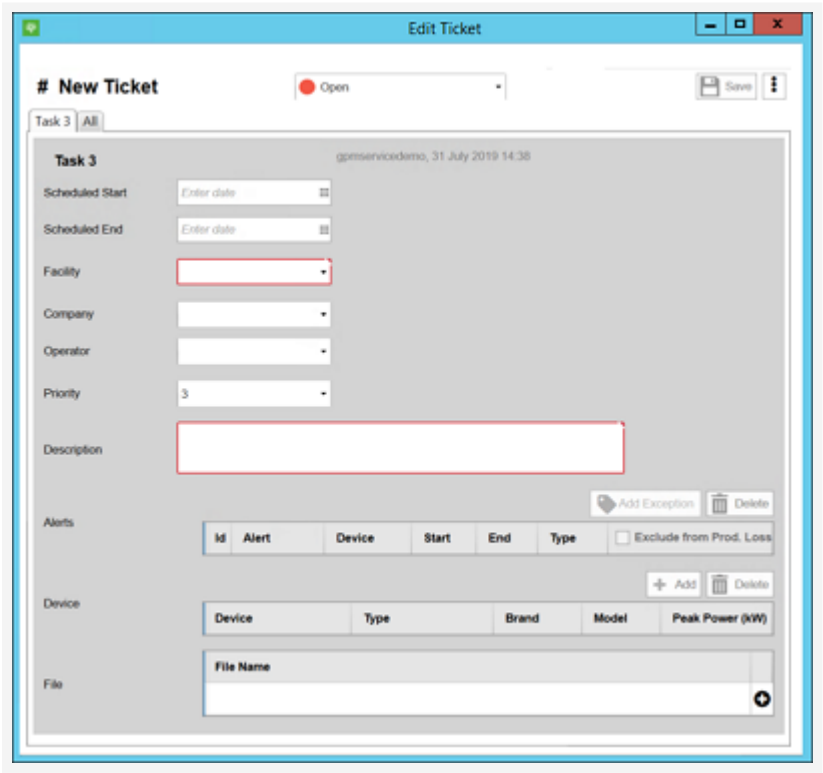
Ticket types



3. In the menu, select **Maintenance**, then select the **GPM Default** template.

Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog



4 In the **Edit Ticket** dialog, enter the information for the ticket:

a **Scheduled Start**: select the date on which the ticket starts.

b **Scheduled End**: select the date on which you expect the ticket to close.

c **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.

d **Company**: select the company that must resolve the issue related to the ticket.

e **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

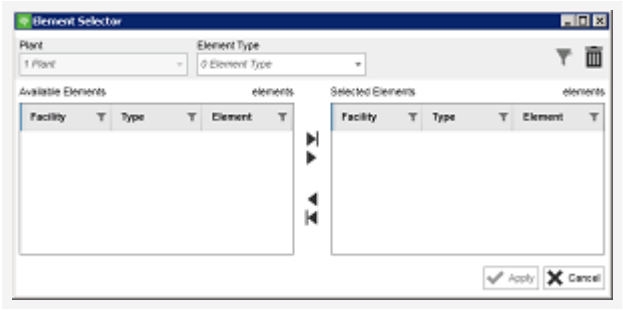
h **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

i (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

5. (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a. Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b. In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c. Click **Apply**.
Result: The element is added to the list.

6. (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).
7. Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

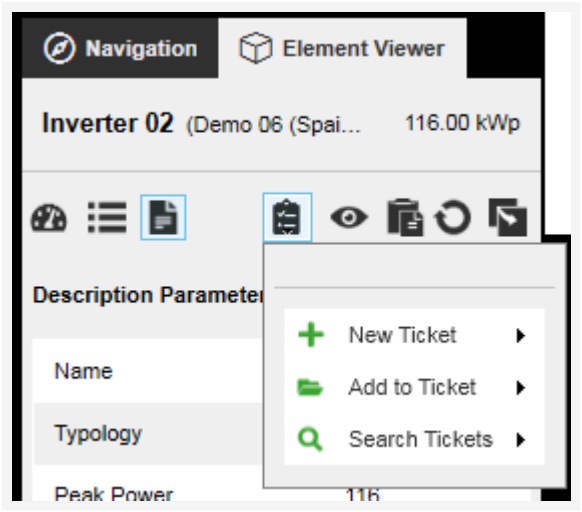
Create data correction tickets from elements

To create a data correction ticket from an element, follow these steps:

1. In the Element viewer module, click the icon.

   Result: The **Ticket** menu opens:

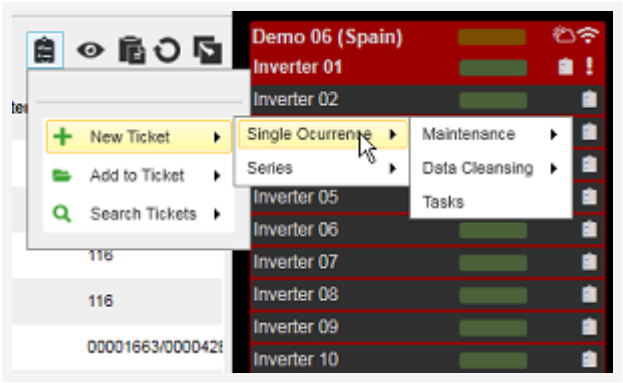
   ### Ticket menu

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.

   Result: The ticket type panel opens:

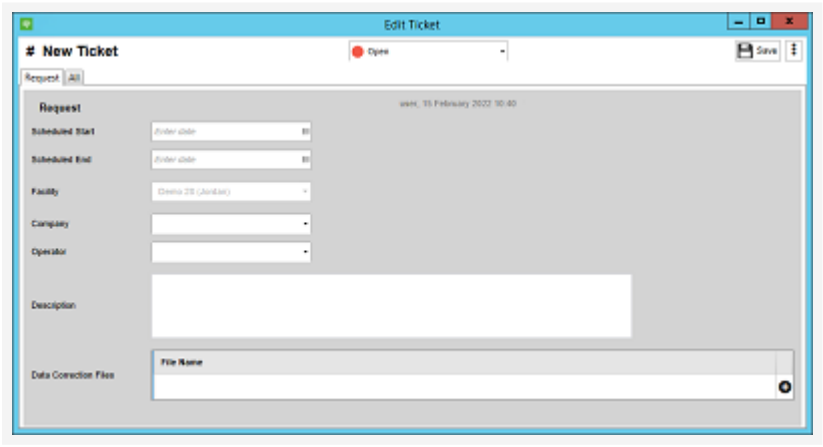
   ### Ticket types

3. In the menu, select **Data Cleansing**, then select **GPM Data Correction**.

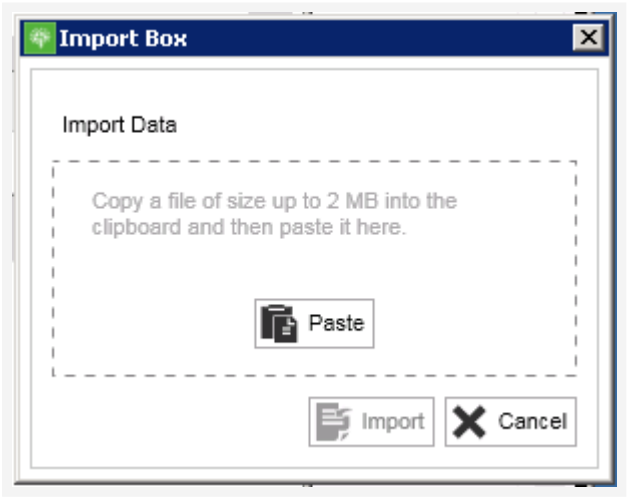
Result: The Edit ticket dialog appears:

Edit ticket dialog (Data cleansing)



4. In the Request section, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. *Description*: enter a description in the text input field.
5. In the Data Correction Files section, click the ⊕ icon to open the Import Box and import an XLS file from which to add the corrected data:

Import Box



ⓘ NOTE: You can only import XLS files.

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

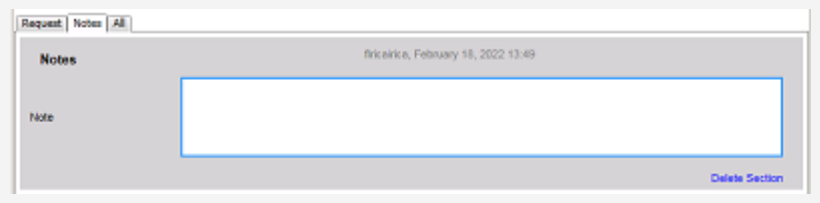
Result: The file is imported to the ticket.

6. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition



b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

7. Click **Save**.

Result

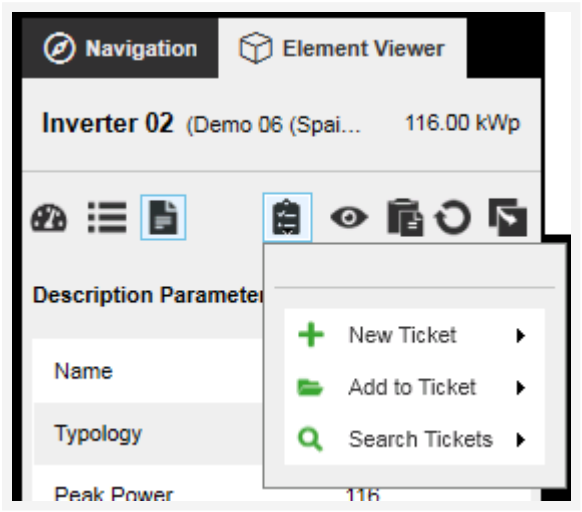
The ticket is created and assigned to the operator you selected.

Create task tickets from elements

To create a task ticket from an element, follow these steps:

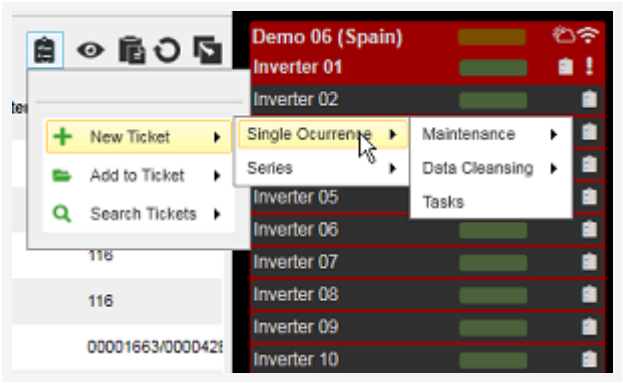
1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

   ### Ticket menu

2. Hover over **+ New Ticket** to open the options panel and select **Single occurrence**.
   Result: The ticket type panel opens:

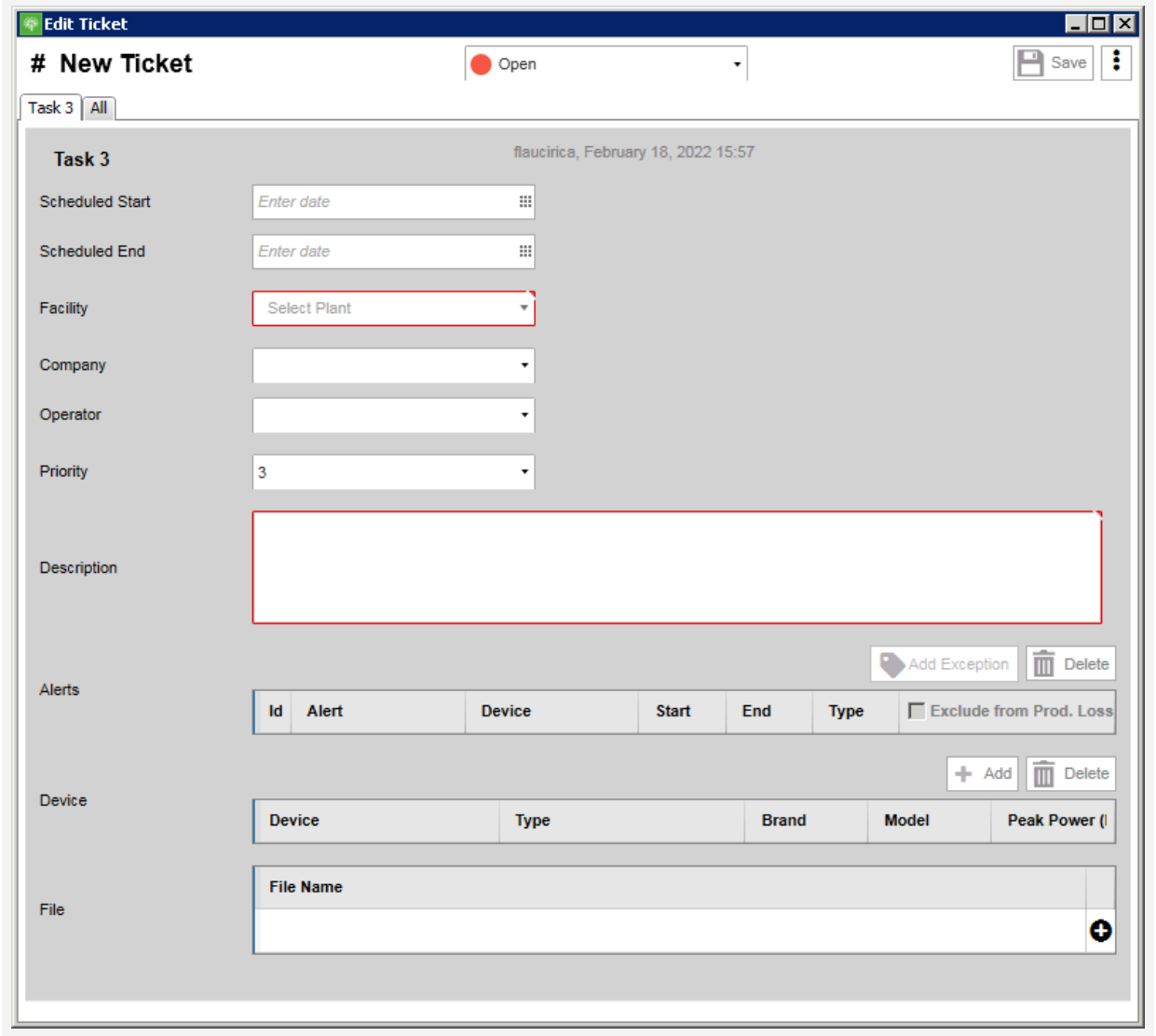
   ### Ticket types

3. In the menu, select **Task**, then select the **GPM Default** template.

Result: The **Edit** ticket dialog appears:

Edit ticket dialog (task)



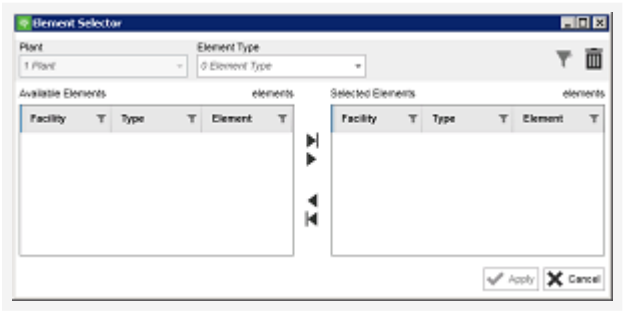
4. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

5 (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b In the **Available Elements** panel select the element you want to add to the ticket, then click the ▶ icon.
OR: Click the ▶| icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c Click **Apply**.
Result: The element is added to the list.

6 (Optional) In the File section, click the ⊕ icon to open the Import Box and import a file.

Import box

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

Result: The file is imported to the ticket.

7. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition

b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

8. Click **Save**.

Result

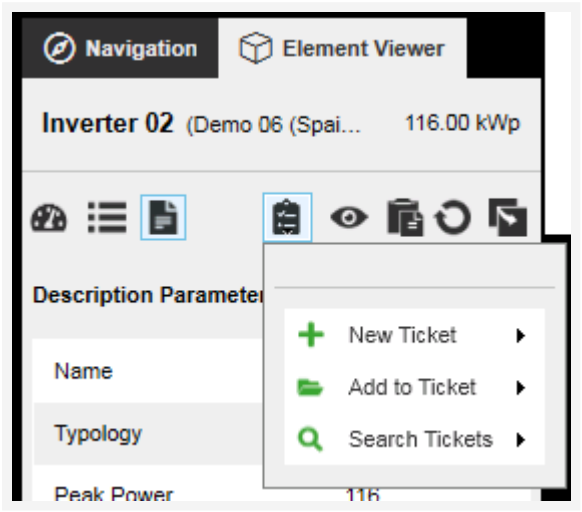
The ticket is created and assigned to the operator you selected.

Add elements to existing tickets

To add an element to an existing ticket, follow these steps:

1. In the Element viewer module, click the icon.
   Result: The **Ticket** menu opens:

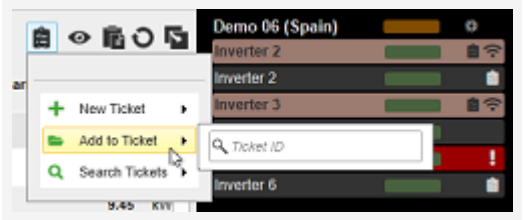
   ### Ticket menu

2. Hover over **Add to Ticket** and enter the *Ticket ID* of an existing ticket (for example, "1543549623"), and press **Enter**.

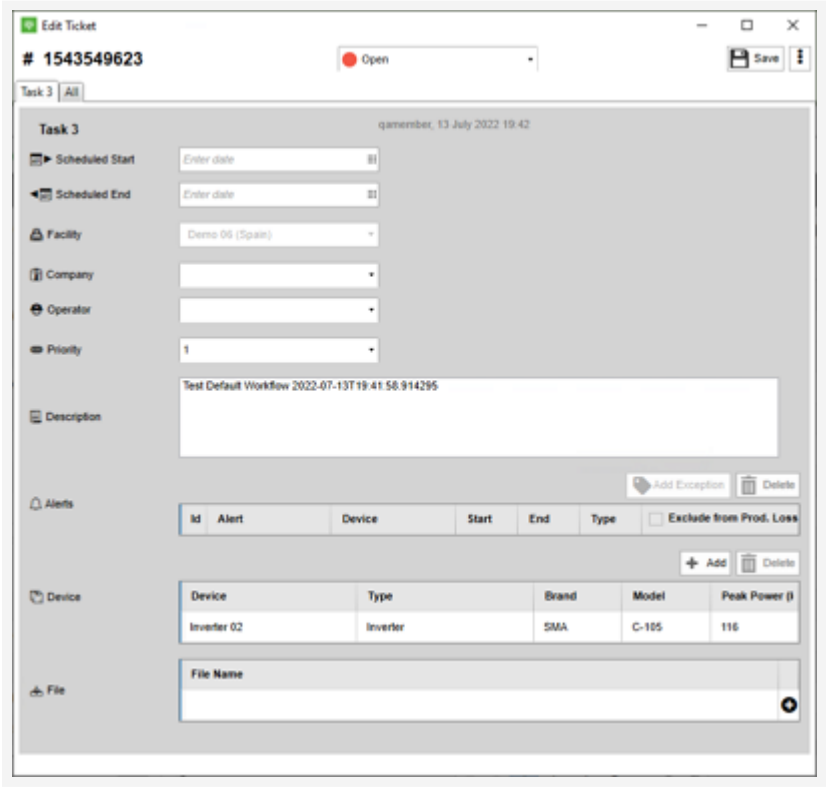
   > REMEMBER: You must enter the ID of an existing ticket. If no ticket exists, follow the instructions to create a new ticket from the Element viewer.

   ### Ticket menu

Result: The **Edit ticket** dialog appears, displaying the selected element in the Element field:

Edit ticket dialog



3. (Optional) Edit any other fields of the ticket you want to change.
4. Click **Save**.

Result

The element is added to the ticket and any other changes are saved.

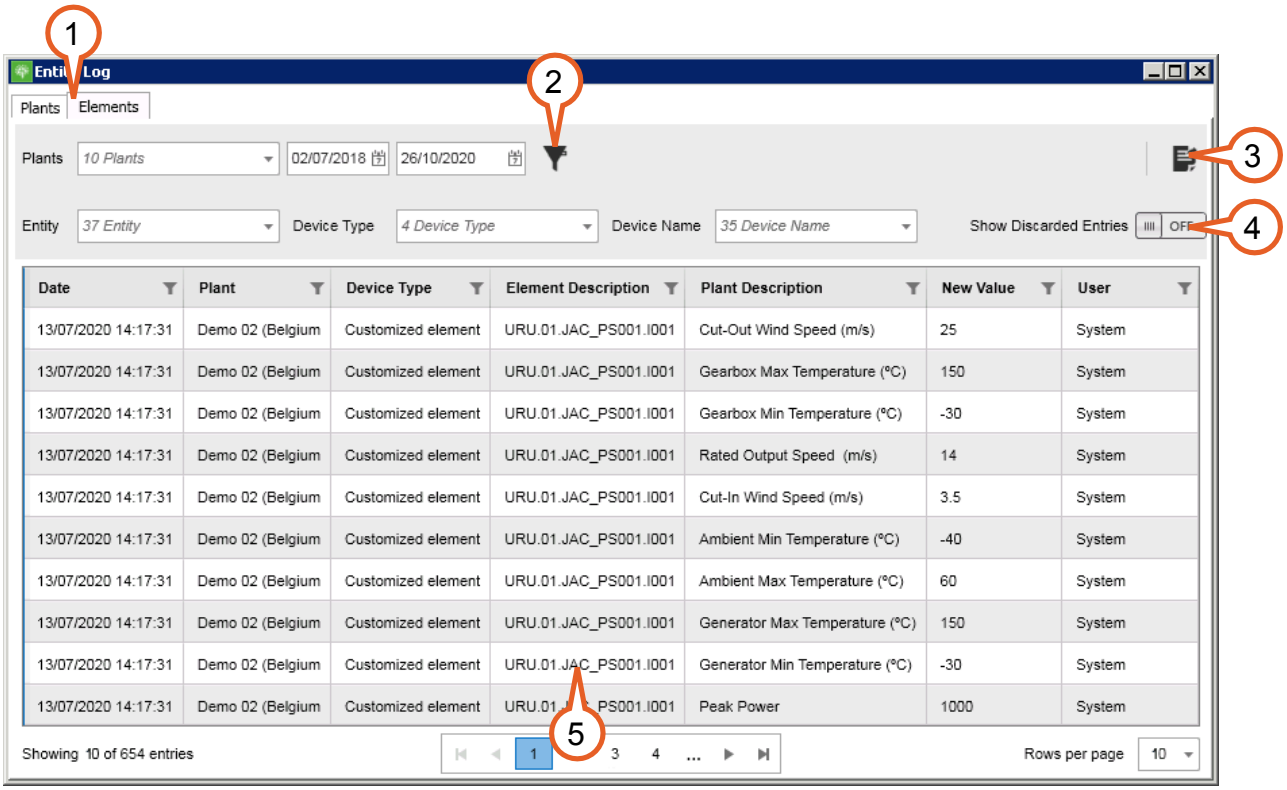
Entity log module

The Entity Log helps you audit and manage the metadata of your plants and devices by tracking changes to values, the timestamp of the change, and the user who made it.

Metadata values are more inclined to human error, because users input them manually. This means that you may want to use the Entity Log to restore an old value or to confirm with a user that the change was intended. Additionally, you can discard values for a more focused browsing and to exclude them when you export this data.

You can access the Entity Log by right-clicking any metadata in the **Info** tab of the Element Viewer and selecting **Open Log**.

Entity Log



1. **Entity tabs**: each available entity type has its own log in a different tab. Click on a tab to display the relevant entity log.
2. **Filter**: the system applies a filter with the criteria of the selected metadata by default. You can edit the filter and click to apply the new filtering criteria.

   Filtering criteria change based on the selected entity
3. **Export**: click to export entries that are visible in the list. For further information, see Export Data to File.

4. **Discard entities**: toggle on or off to to display or hide discarded values so that they are not exported. This is useful because the export feature only exports visible entries.
5. Log: tracks all the changes made to the values. If a value was changed by mistake, you can right-click the entry and discard it, so that it will be excluded from the export. You can restore discarded values at any time.

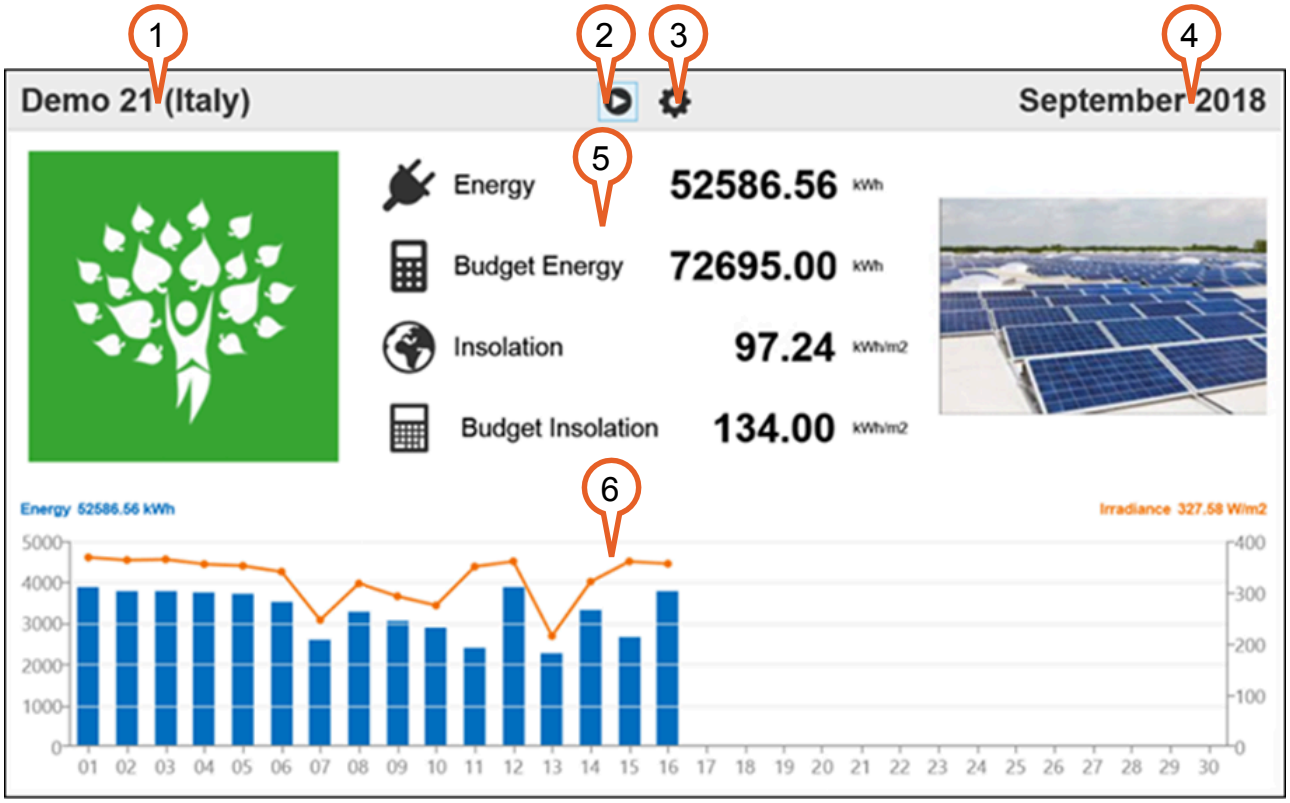
Kiosk View

The Kiosk View module allows you to display a slideshow with cards containing the main KPIs of the plants in your portfolio. For example, you can open the Kiosk View module in a cyclic playback mode in a public display, so that the most up-to-date values can be monitored at any time.

You can customize the Kiosk View module with up to four KPIs and one chart displaying two parameters. Additionally, it can display the logo of your Company logo and another image, such as a photo of the plant. For more information, see the Slide Configuration section.

To access the Kiosk View module, click the icon on the upper bar, then click the icon.

Kiosk View



1. **Plant name**: click to select another plant from the drop-down list

   ⓘ NOTE: Selecting another plant stops the slideshow.

2. **Resume**: click to resume the slideshow.
3. **Settings**: click to open the Kiosk View module settings.
4. **Period**: click to customize the period of the data on display. Available options are

day, month, year, and lifetime.

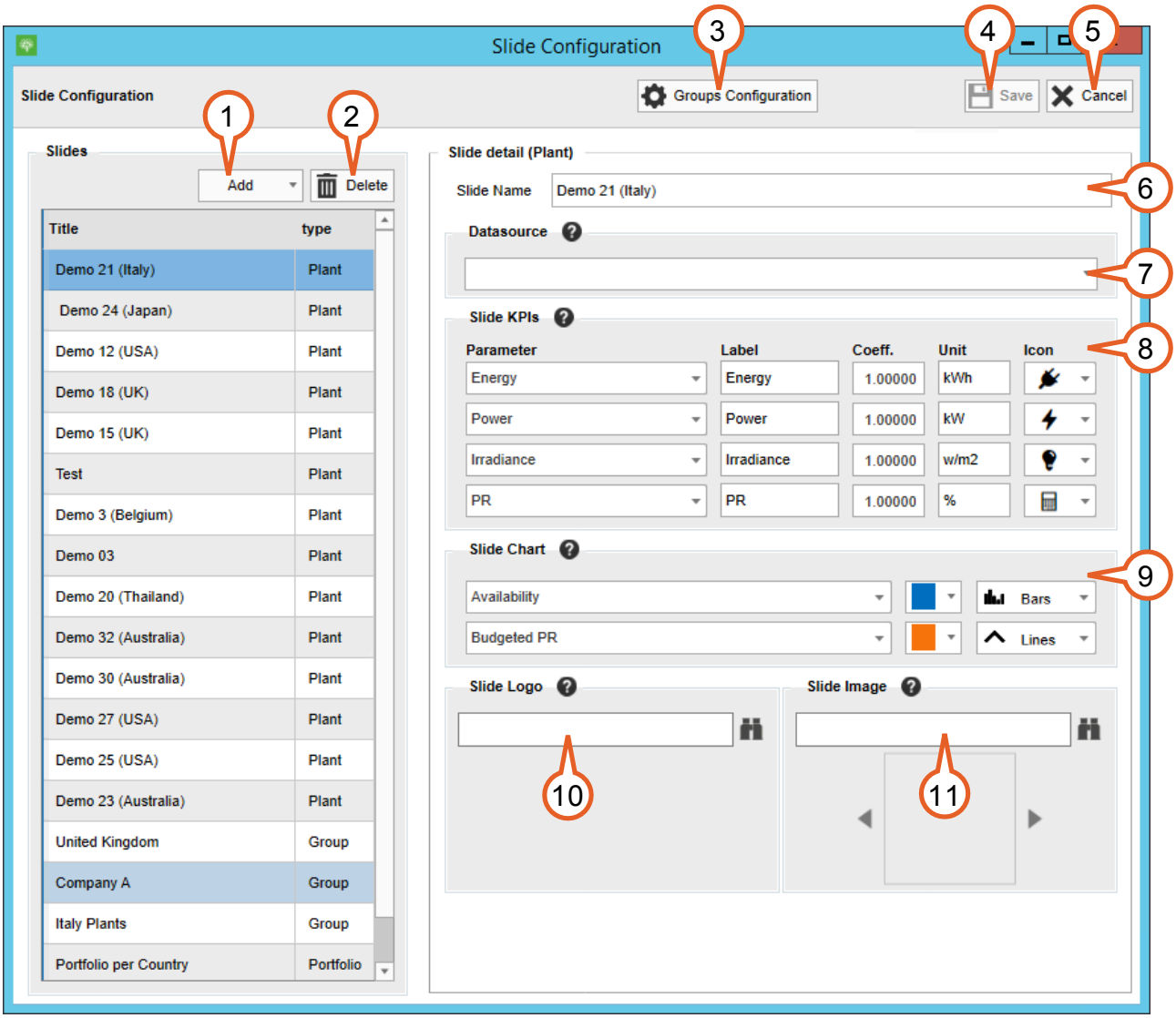
5. KPIs: displays pre-configured KPI values for the selected period.
6. Chart: displays a chart produced by comparing two parameters.

Setting	Decription
Slides	Select the plants to display in the slideshow. Click and drag a plant to sort the order in which they appear.
Included time periods	Enable a checkbox to display slides for the selected time period. You can choose multiple options. The available options are: **Day**, **Month**, **Year** and **Lifetime**.
Play order	Select the order in which to display the slides and the time periods.
Slide preferences	Click and drag the slider to define the time between slides (for example, 10 seconds). Click the **Transition effect between frames** menu and select an option for transitions between slides.

Slide configuration

You can customize the configuration of the slides for one or more plants by clicking the **Slide Configuration** button in the Settings screen.

Slide Configuration



1. **Add**: click to add a new slide and select the type of data you want to include in it. The available options are **Group**, **Plant** and **Portfolio**.
2. **Delete**: click to delete the selected slide from the slideshow.
3. **Groups configuration**: click to create and configure groups of plants you can then add to the slides.
4. **Save**: click to save your changes.
5. **Cancel**: click to discard your changes.

6. *Slide name*: enter a name for the slide. By default, the slide name is the same as the plant name.
7. **Datasource**: click to select a plant or a group of plants from the drop-down menu.
8. Slide KPIs: click each field to define the data you want to display, and how to display it:
   - **Parameter**: click to select a parameter from the drop-down menu.
   - *Label*: enter a name for the label you want to assign to the parameter.
   - *Coefficient*: enter a coefficient for the parameter.
   - *Unit*: enter a unit for the parameter.
   - **Icon**: click to select an icon for the parameter from the drop-down menu.
9. Slide chart: click each field to configure the chart that appears on the slide:
   - **Parameters**: click to select a parameter from the drop-down menu.
   - **Parameter color**: click to select a color for the parameter.
   - **Graph type**: click to select the type of graph you want to display for the parameter.
10. Slide logo: click the icon to import a corporate image to display on the slide (for example, your company's logo).
11. Slide image: click the arrows to browse between available images and select one to display on the slide, or click the icon to upload a new image.

Linear Chart Viewer

The Linear Chart Viewer module is a tool that allows you to create queries and analyze your portfolio's performance by creating charts that display parameter values at specific moments in time. The results of your queries can be saved for further use or exported to other views.

ⓘ BEST PRACTICE: You can open multiple Linear Chart Viewer module windows, but you can only add queries to the active window. The active window has the icon.

Linear Chart Viewer module

1. Chart: displays the results of your query.

   Right-click a point of the chart to display the context menu. From the context menu, you can click **Display Raw Values** to display the data in the Live Viewer module.
2. Query period: click the icon to expand the menu and customize the data

granularity and time range. For information, see the Query Period section below.

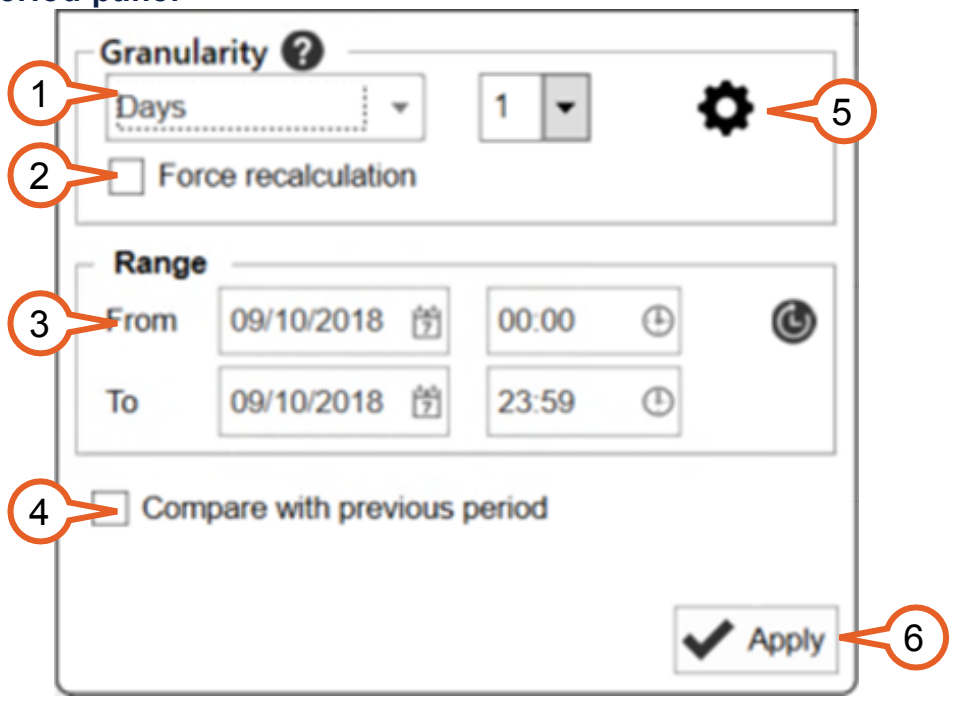
3. Add parameters:click the icon to expand the menu and add parameters to the table. For more information, see the Add Parameters section. below.
4. **Action buttons**: click to perform quick actions:
   - Toggle zoom on and off. When zoom is enabled, use the mouse scroll or drag and drop the area of the chart that you want to zoom.
   - Save the current query as a favorite.
   - Load a favorite query.
   - Display the selected parameters in the Linear Chart Viewer module.
   - Display the parameters in the Live Viewer module. This option is only available in SCADA environments.
   - Export data to the Live Viewer module.
   - Export the query to the clipboard or to a file. For further information, see Export Data to File.
   - Toggle the window lock on and off. When you lock a window, you cannot add further parameters to it from other parts of the application.
5. Current parameters: click the icon to expand the menu and manage current parameters. You can change the display name and color for a parameter. You can also decompose parameters. For more information, see the Current Parameters section below.
6. Legend: displays the results of your query as a table. You can hide data from the chart and customize the appearance of single parameters. For further information, see the Legend section below.
7. **Period browser**: browse the time range using the arrows. The time range displayed is based on the query period that you selected.
8. **Unlock time bar**: click to enable the time bar usage on the chart.

   When the time bar is active, you can place your cursor over a point on the chart to display a tooltip with further information. When you click that point on the chart, the time bar is fixed, and you can analyze the data from that point on the Legend. For further information, see the Legend section below.

Query Period

The Query Period panel lets you customize the data granularity and time range of your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Query Period panel



1. **Data granularity**: leave the default data granularity setting or select a data granularity from the drop-down list.

   Select a data granularity value if the selected granularity requires it.

2. **Force recalculation**: click to manually force a recalculation of the data if the values have changed you do not want to wait for the next automatic recalculation to see them. The system automatically calculates values at recurring intervals and stores them in the database for faster access.

   This option is only available when the selected granularity is **Days**, **Months**, or **Years**.

   ⓘ NOTE: Recalculations may take several minutes.

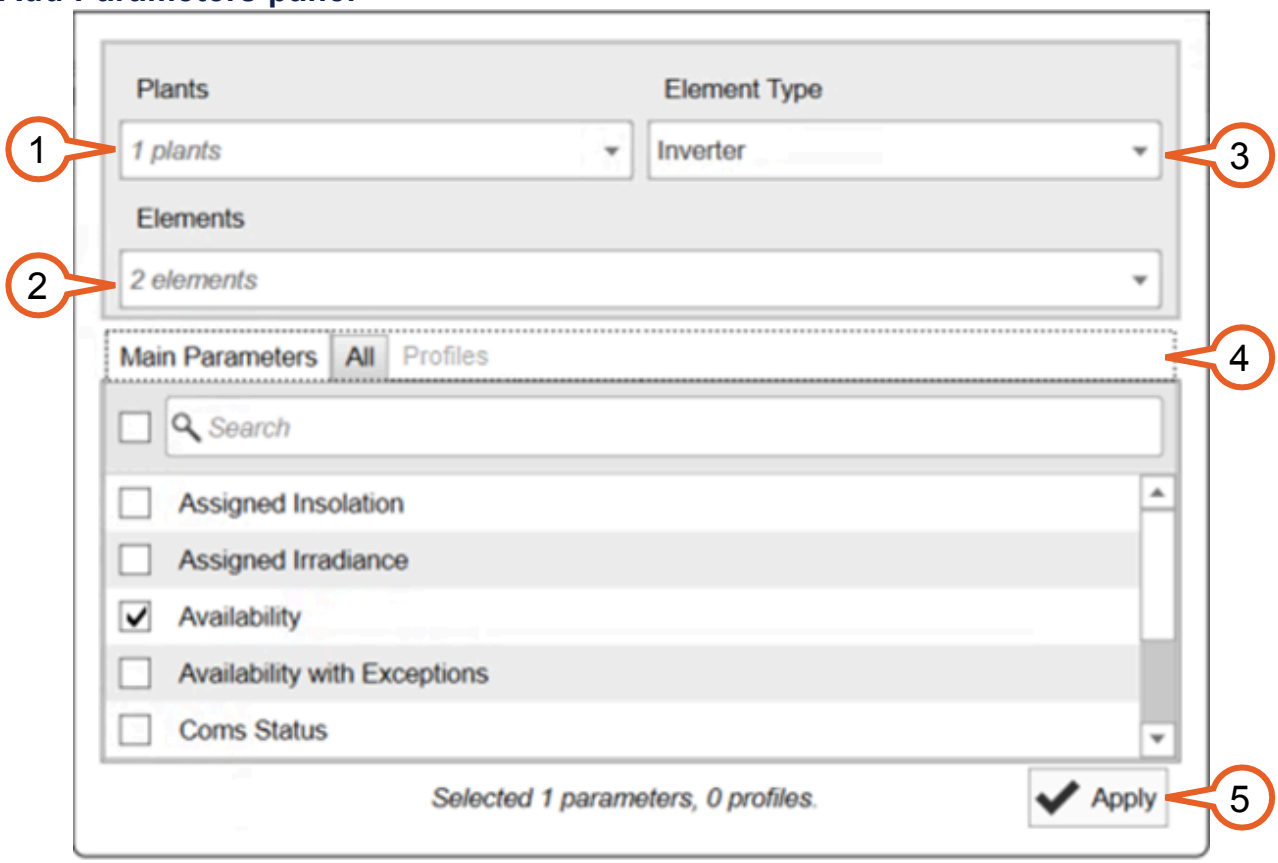
3. **Date and time range**: select the date and time range using the calendar picker. Click the ⏱ icon to set the range to the default period.
4. **Compare with previous period**: select the checkbox if you want the chart to include data from the previous period to the one you selected. For example, you can select today as the period and select this checkbox to include yesterday's data.

5. **Data granularity settings**: click to access advanced settings for data granularity.
6. **Apply**: click to apply your settings to the query.

Add Parameters

The Add Parameters panel lest you add parameters to your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Add Parameters panel



1. **Plants**: click to select one or more plants from the drop-down list. You can use the *Search* field to refine the drop-down list results.
2. **Elements**: click to select one or more element from the drop-down list.

   ⓘ NOTE: You can only select elements after selecting a single plant. If you select multiple plants, this option is disabled.

3. **Element type**: click to select one or more element type from the drop-down list.

   ⓘ NOTE: You can only select element types after selecting multiple plants.

4. Parameters: click to select one or more parameters or series from one of the tabs. You can use the *Search* field to refine the drop-down list results.
   - **Main Parameters**: lists the most relevant parameters that are configured for

the selected element.

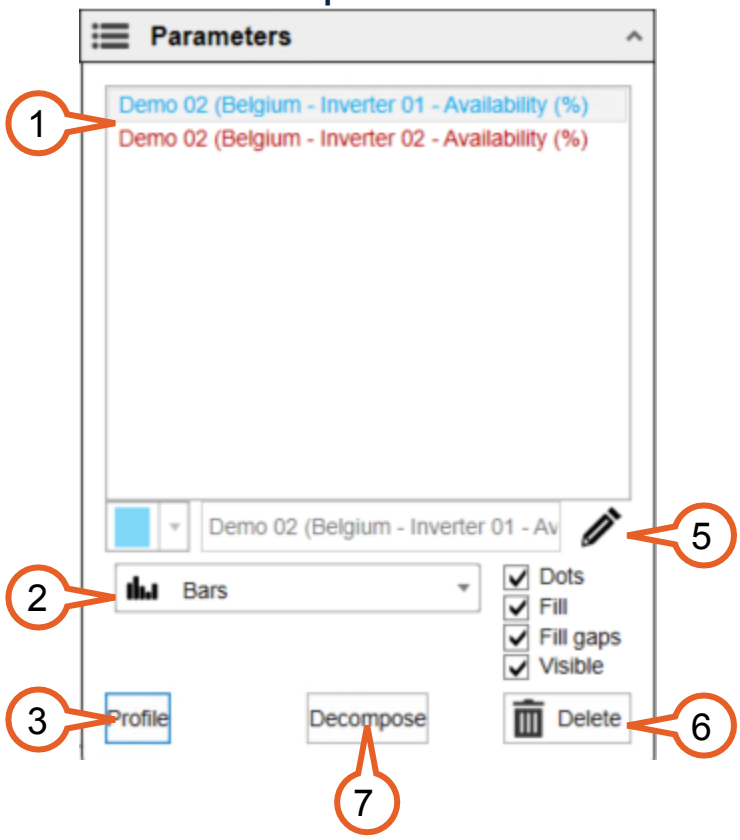
- **All**: lists all the parameters that are retrieved from the selected element.
- **Profiles**: lists all the saved profiles.

5. **Apply**: click to apply your settings to the query.

Current Parameters

The Current Parameters panel lets you the manage parameters included in a query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons. You can also change the display name and the color of a parameter.

Current Parameters panel



1. Parameters: lists the parameters currently displayed on the table. Select one or more parameters to customize their appearance. To select multiple parameters, use CTRL+Click.
2. Chart format: select the chart format from the drop-down list:
   - ⌃ **Lines**: display data as lines.
   - ⌒ **Curves**: display data as curves.
   - **Bars**: display data as bars.
   - **Points**: display data as points.
3. **Save as profile**: click to save the query as a profile and use it as a reference in other charts. You can select saved profiles in the **Profiles** tab of the Add Parameters panel.
4. **Edit**: click to enable editing for the selected parameter. You can customize the

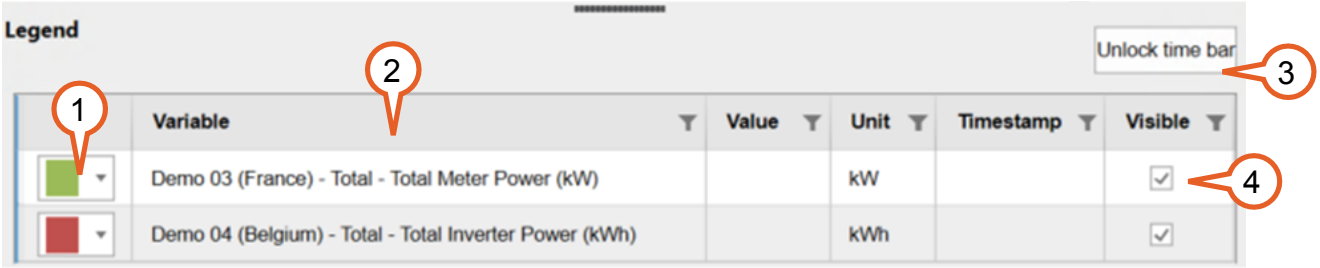
background color and the column heading text. The default text has the plant name, the element name, and the parameter name.

5. Parameter format: select the checkboxes to customize the parameter appearance:
   - **Points**: display dots on the chart events.
   - **Fill**: fill the chart area with a color.
   - **Fill gaps**: connect points even if there is no data.
   - **Visible**: display the parameter.
6. **Delete**: click to delete the selected parameters from the chart.
7. **Breakdown**: click to add all the factors that make up the selected parameter to the query. Factors can be other parameters or data sources.

Legend

The legend panel provides quick access to the chart legend and displays the results of your queries as a table. You can filter and sort the data on display and customize the parameter colors.

Legend panel



1. **Parameter color**: click to select the display color of a parameter from the drop-down list.
2. Table: displays the plant, device and parameter names; the parameter values; the unit; and the time stamp when the value was retrieved.
3. **Unlock time bar**: when the time bar is active, you can place your cursor over a point on the chart to display a tooltip with further information. When you click that point on the chart, the time bar is fixed, and you can analyze the data from that point.
4. **Visible column**: toggle a checkbox to display or hide the parameter from the chart.

Live viewer module

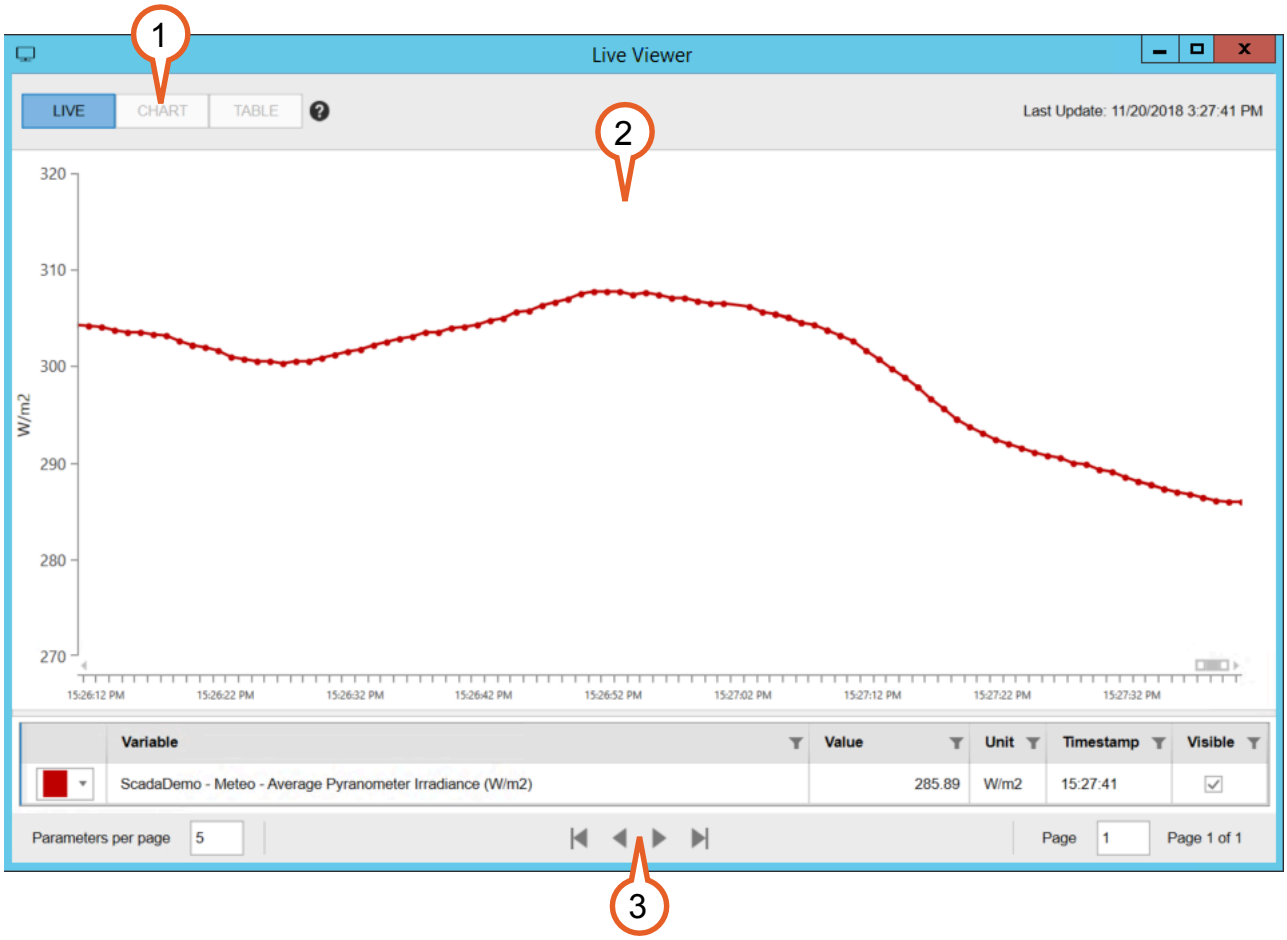
The Live Viewer module is a tool that displays monitored raw data in real time, allowing you to analyze the performance of your portfolio. You can add queries to the Live Viewer module from other areas of the user interface and export the queries created in the Live Viewer module.

There are three display modes for data. Click on each section in the list to see more information:

- Live
- Chart
- Table

You can access the Live Viewer module by clicking the icon next to any parameter in the **Live Values** tab of the Element Viewer module.

Live Viewer



1. Display mode: click to switch between different display modes:
    - **Live**

- **Chart**
- **Table**

ⓘ NOTE: Only the Live mode displays real-time values. Chart and Table modes display monitored values.

2. Data visualization: display the data based on the selected mode.
3. **Pagination**: control the number of columns to display on each page and browse the pages using the arrows or the text input field.

Live tab

The Live tab displays real-time values in a chart format and allows you to analyze data in real time.

Live tab

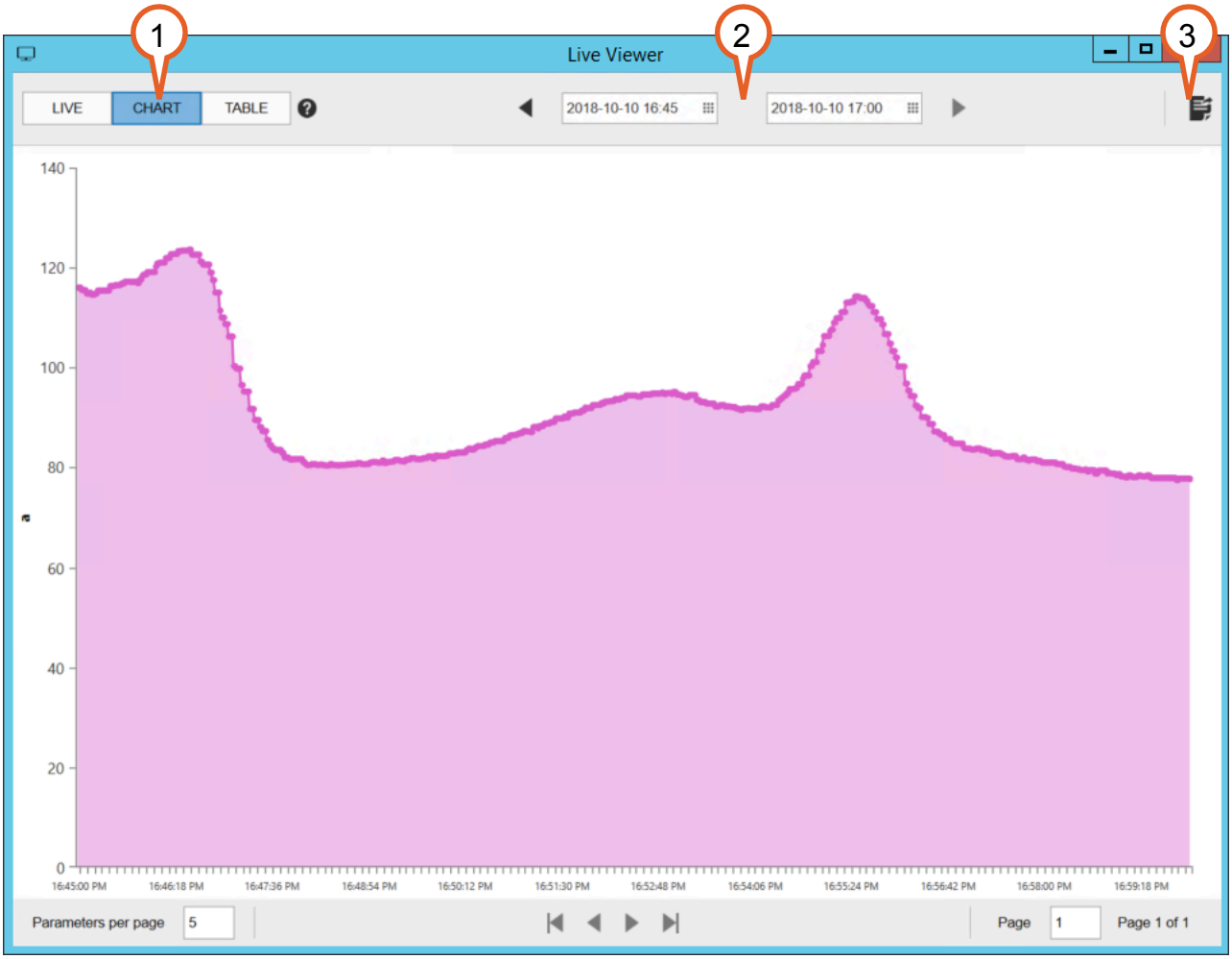
1. Chart: displays a chart of the real-time values. Drag and drop on the chart to zoom in.
2. Time stamp: displays the time stamp of the last update.
3. Table: displays the plant, device and parameter name; the parameter value; the unit; and the time stamp when the value was retrieved. Click the ▼ icon on any column header for advanced filtering. For further information, see Advanced Filters.
4. **Parameter color**: select the display color of a parameter from the drop-down list.
5. **Visible column**: toggle the checkbox to display or hide the parameter from the chart.

Chart tab

The Chart tab displays non-live raw values in a chart format and allows you to analyze data during a specific time range.

Raw values are values as they are received by the system, without any further calculation.

Chart tab



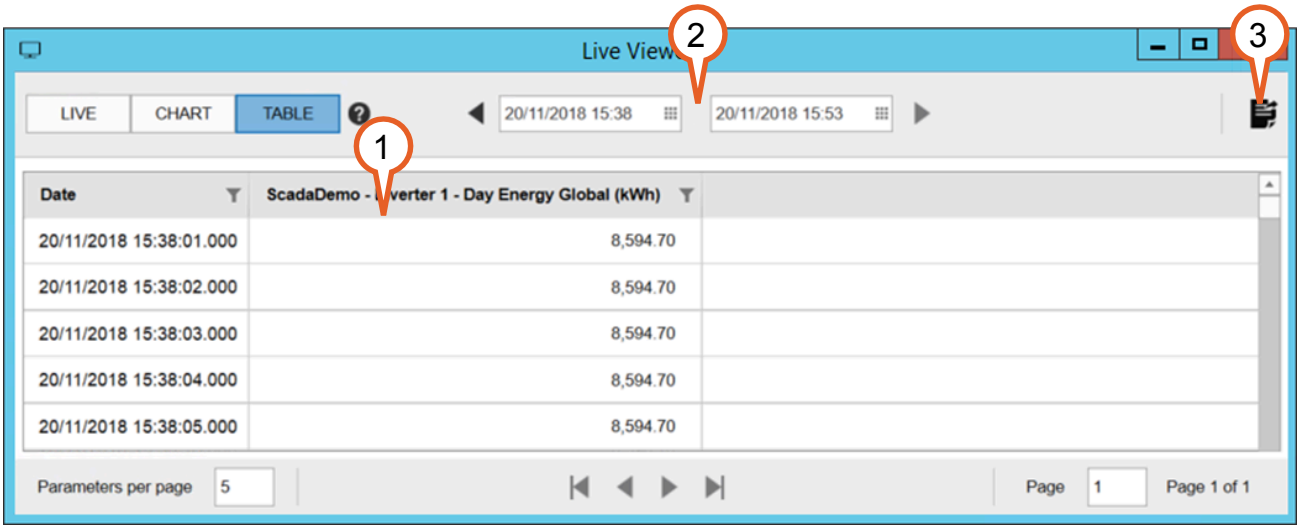
1. Chart: displays a chart with a set of values retrieved at recurring intervals.

   Place your cursor over a dot to display a tooltip with details about the element from which the data was retrieved, the raw value, and the time stamp.
2. **Date range picker**: change the range date and time. The maximum range allowed is the interval at which data retrieved.
3. **Export**: click to export the chart as a JPEG image or as an Excel file. For further information, see Export Data to File.

Table tab

The Table tab displays non-live raw values in a table format and allows you to analyze data during a specific time range.

Table tab



1. Table: displays a table with a set of values retrieved at recurring intervals. Click the column header to sort the table by the values of that column. Click the icon on any column header for advanced filtering. For further information on advanced filtering, see Advanced Filters.
2. **Date range picker**: change the range date and time. The maximum range allowed is the interval at which data retrieved.
3. **Export**: click to export the chart as a JPEG image or as an Excel file. For further information, see Export Data to File.

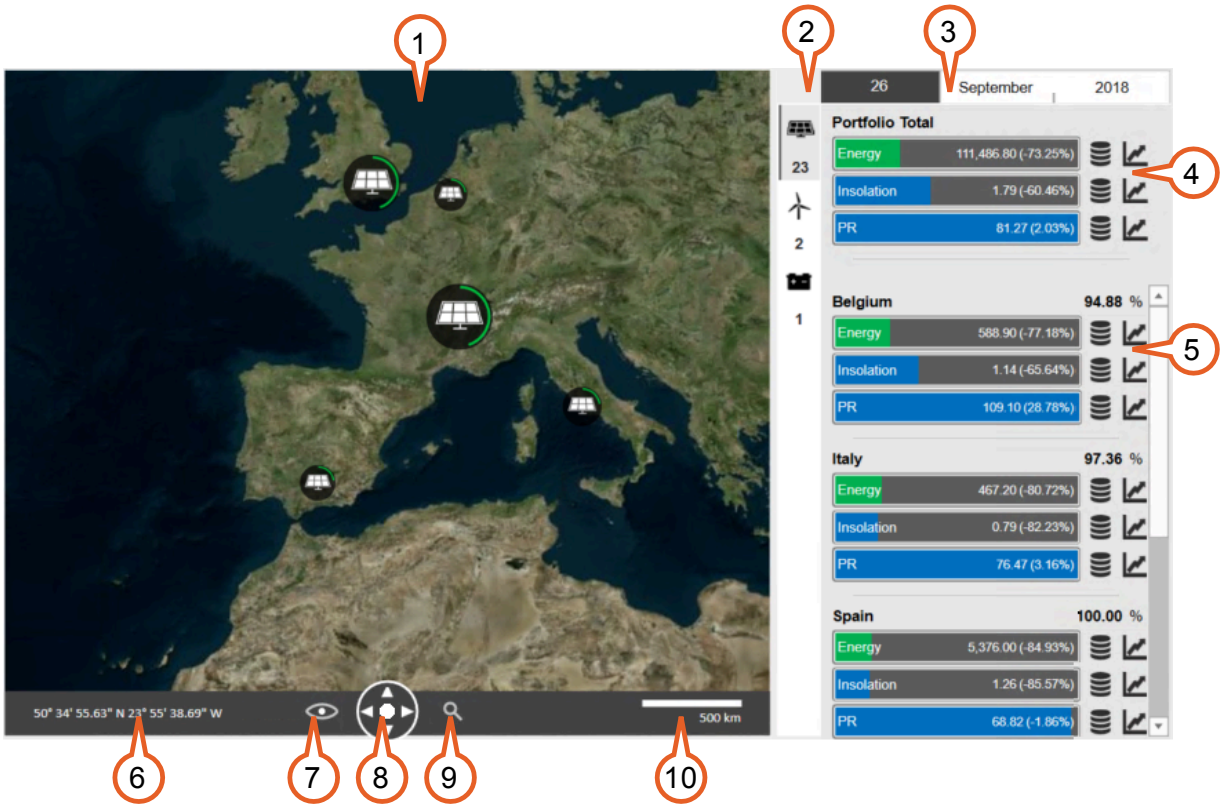
Map

The Map module provides a geographical overview of your portfolio by placing each plant on a map based on their geo-location. The Map module provides KPIs to monitor the portfolio performance as well as the specific performance of each plant.

The Map module allows you to navigate and monitor your portfolio by layers, providing information at different degrees of detail when you zoom in or out of the map.

The Data Panel located on the right side provides further tools to monitor the performance of your portfolio by location, technology, and time stamp.

Map



1. Map: monitor your plants or clusters based on their geo-location. Drag and drop the map to browse it.
2. Technology: select a technology to display plants with a specific technology in the Plant KPIs area. This option is available when you monitor a multi-technology cluster.
3. **Date selector**: select the date range for which to display the KPIs. You can select the current day, month or year.

4. **Portolio KPIs**: displays the total KPIs of the plants currently displayed on the map.
   Click the icon to display the KPIs in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.
5. Plant KPIs: displays up to four plant-specific KPIs for plants that are currently visible on the map.
   Click the icon to display the KPIs in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.
6. Geolocation: displays the coordinates of the last point clicked on the map.
7. **Map options**: click the icon to customize the map:
   - **Road Map**: display the map as a road map.
   - **Satellite**: display the map as a seen from a satellite.
   - **Terrain**: display the map with terrain information.
   - **Show labels**: toggle to display or hide city and road labels.
8. **Navigation**: click the arrows to move the map to the four sides or click the central button to set the zoom to minimum.
9. **Zoom**: click and select the map zoom. You can also use your mouse scroll button to zoom in and out of the map.

   There are three levels of zoom:
   - **Default**: includes all the plants in the portfolio.
   - **Country**: displays a single icon for each country, with information on the overall performance for all the plants in it.
   - **Region**: displays one icon per plant with information on its performance.
10. Scale: displays the current scale of the map.

Entities in the Map module

The Map module has dynamic icons that reflect the performance of clusters, plants and devices. Each icon has a progress bar to indicate its power production, expressed as a percentage of its peak power.

Icon	Type
	General
	Solar
	Wind
	Storage
	Biogas
	Gas
	Cluster

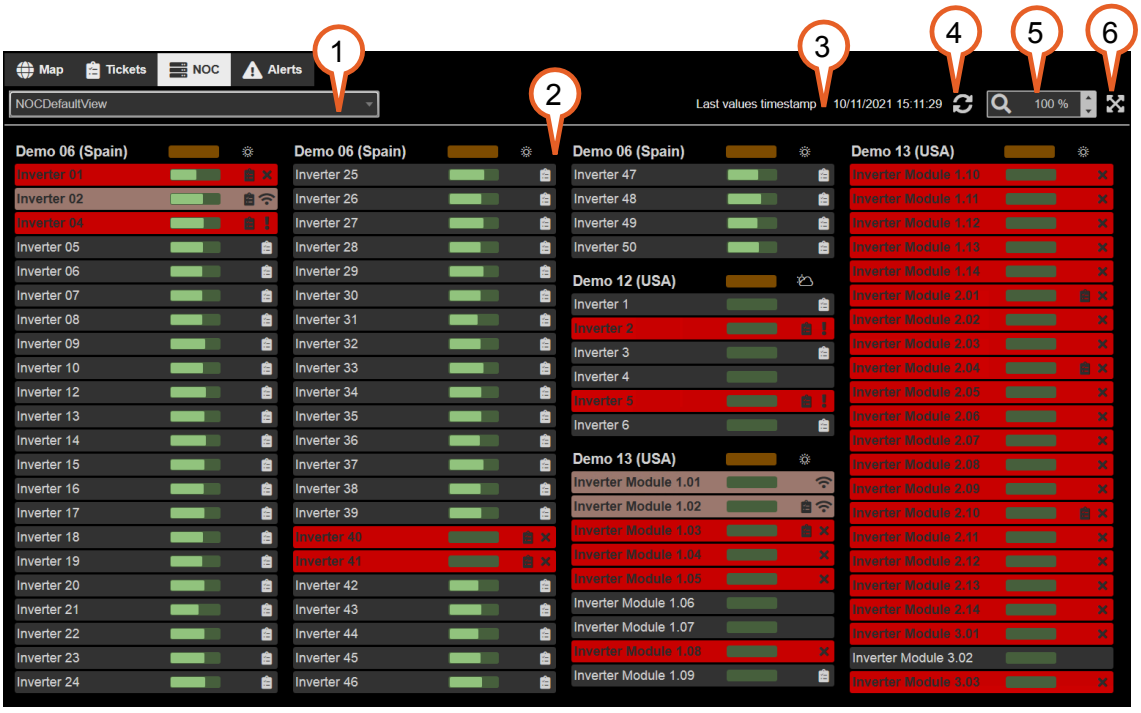
Network Operating Center (NOC)

The NOC (Network Operating Center) module is a dashboard designed for operations and maintenance users. It makes the portfolio easy to monitor because it condenses its relevant information in one area.

The NOC displays cards with information that refreshes automatically at regular intervals. By default, the NOC displays the cards for all the plants in your portfolio, with its corresponding elements.

The heading and each row of a card appear in the color that corresponds to the most relevant alarm for the corresponding plant and device.

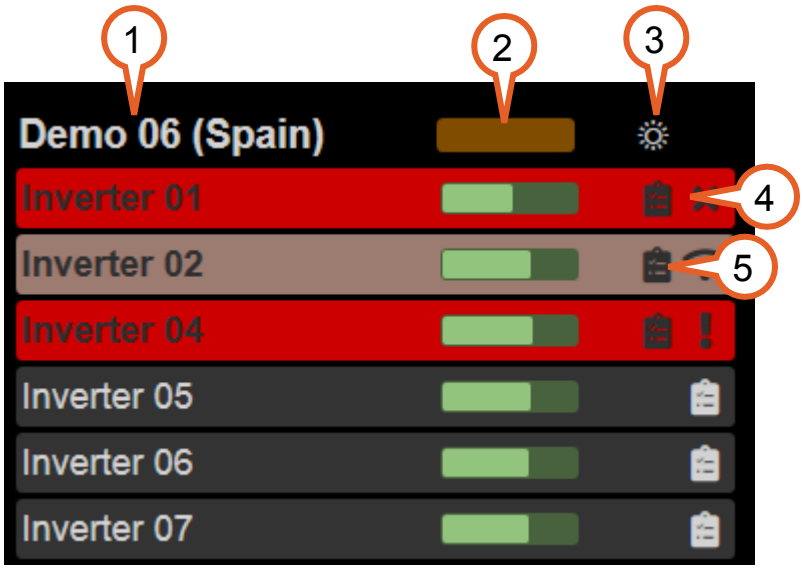
Network Operating Center (NOC)



1. **View mode**: select one of the available views from the drop-down menu. You can open views in a separate window by clicking the icon next to an option.
2. Cards area: displays the selected view. Views are pre-configured and display the information as cards that auto-resize to fit your screen. Each card corresponds to a plant and displays a set of child devices.
3. Timestamp: provides the time at which the values displayed on the NOC module were last updated.

4. **Refresh**: click to manually refresh the values displayed on the Cards Area.
5. Zoom: select the level of zoom to apply to the Cards Area. You can use the arrows to increase the zoom, decrease it, or type the zoom percentage manually.
6. **Auto-fit**: click to automatically resize the cards to fit your screen.

NOC card



1. **Plant and element names**: displays the names of the plant and its elements.

   Click on a name to display further details in the Element Viewer.
2. **Performance bar**: provide a quick view of the performance of the plant or device. The system calculates the level of performance by comparing the real value received from the plant with a predefined reference value.

   Place your cursor on the bar to display the performance details and click the bar to display the performance details in the Linear Chart Viewer module.
3. **Weather**: displays the current weather at the plant. Place your cursor on the icon for further information.

   The weather is retrieved using the plant geo-position from a GPM partner weather service.
4. **Alarms**: displays the icon of the most relevant alarm, if any.

   Click the icon to display further information about the alarm in the Alarms Information panel.

5. **Ticket status**: Displays ticket icon when there is any active ticket related to the device.

   Click the icon to open the Edit Work Order dialog.

   Place your cursor on the icon to display the ticket description.

Permissions module

The Permissions module allows you to manage the permissions that restrict access to entities in GPM SCADA. Permissions work by using tags to link roles and the entities (for example, datasources) to which they have access. When you assign a tag to an entity, it becomes accessible only to the user roles which also have the tag assigned to them.

> ⓘ NOTE: The permissions you assign to the user roles also affect the access that users have in GPM Horizon.

The Permissions module has two tabs:

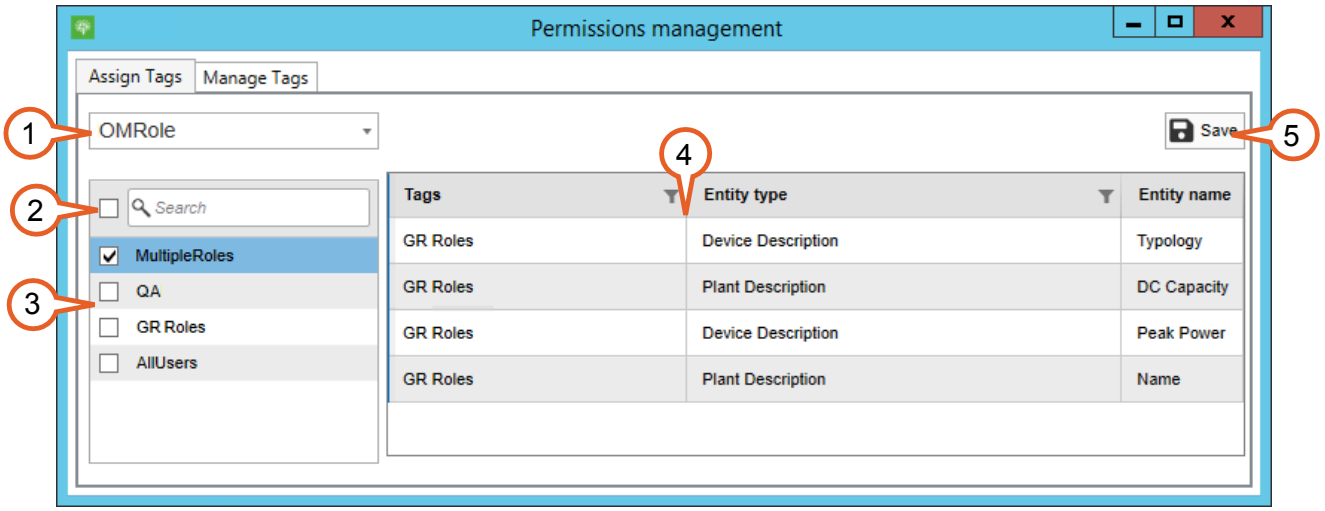
- Assign tags, were you assign tags to user roles.
- Manage tags, where you create, edit and delete tags.

To access the Permissions module, click the ⚙ icon on the Upper Bar.

ⓘ NOTE: For more information about the system of permissions, see the article on Permissions. For more information about working with tags, see Manage permissions.

Assign Tabs

Assign Tags tab

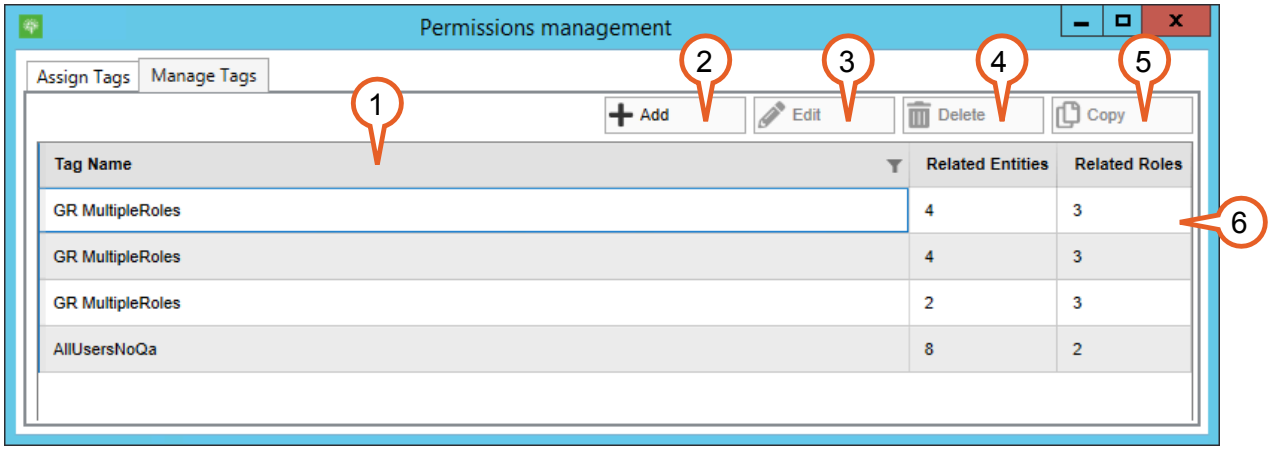


1. **Roles**: click the drop-down menu to see the available user roles.
2. *Search bar*: enter text to narrow down the available tags.
3. **Tags**: list of available tags you can assign to roles.

   Click the checkbox next to the name of a tag to select it and make changes to it.
4. Tags table: lists the tags assigned to the selected role.
5. **Save button**: click to save changes to roles and tags.

Manage Tabs

Manage Tags tab



1. **Tags table**: displays all the tags in your system.

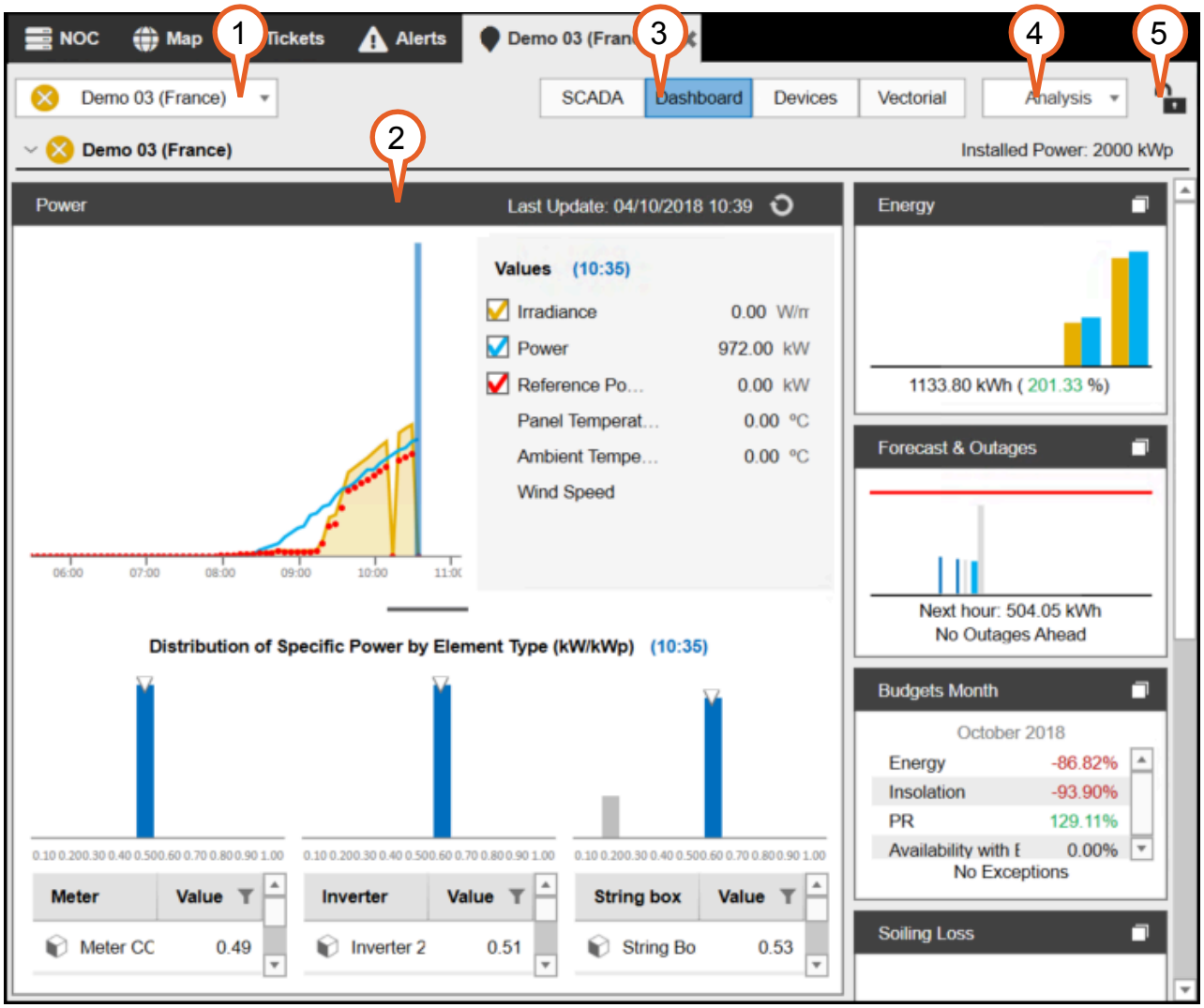
   Click on a tag to select it in order to edit, delete, or copy it.
2. **Add**: click to add a new tag.
3. **Edit**: click to edit a selected tag.
4. **Delete**: click to delete a selected tag.
5. **Copy**: click to copy a selected tag.
6. Total links: displays the total numbers of entities and roles related to the tag.

Plant module

The Plant module displays as a tab of the Content Area and gathers all plant-related tools in the same interface, allowing you to quickly navigate, monitor, and manage your portfolio.

To access the module, click the icon next to a plant or double-click a plant on the Navigation panel. Alternatively, you can right-click the plant and click **Select Plant** on the context menu.

The Plant module is organized in tabs located on the top-right area. Each tab allows you to perform specific monitoring and management tasks.



1. **Plant selection**: click to open the drop-down menu and switch between the plants in your portfolio.
2. Information area: displays the content of the selected tab.
3. **Tabs**: click switch to other plant-related modules:

- Plant Dashboard
- Vectorial Layout
- SCADA Layout
- Plant Elements

4. **Custom queries**: click to open a custom chart associated to the selected plant to open it in a separate window.
5. **Lock button**: click to lock the panel. When the panel is locked and you open a plant from the Navigation panel, its Plant module opens in a new panel.

> ⓘ BEST PRACTICE: You can open up to five Plant modules.

Plant Dashboard module

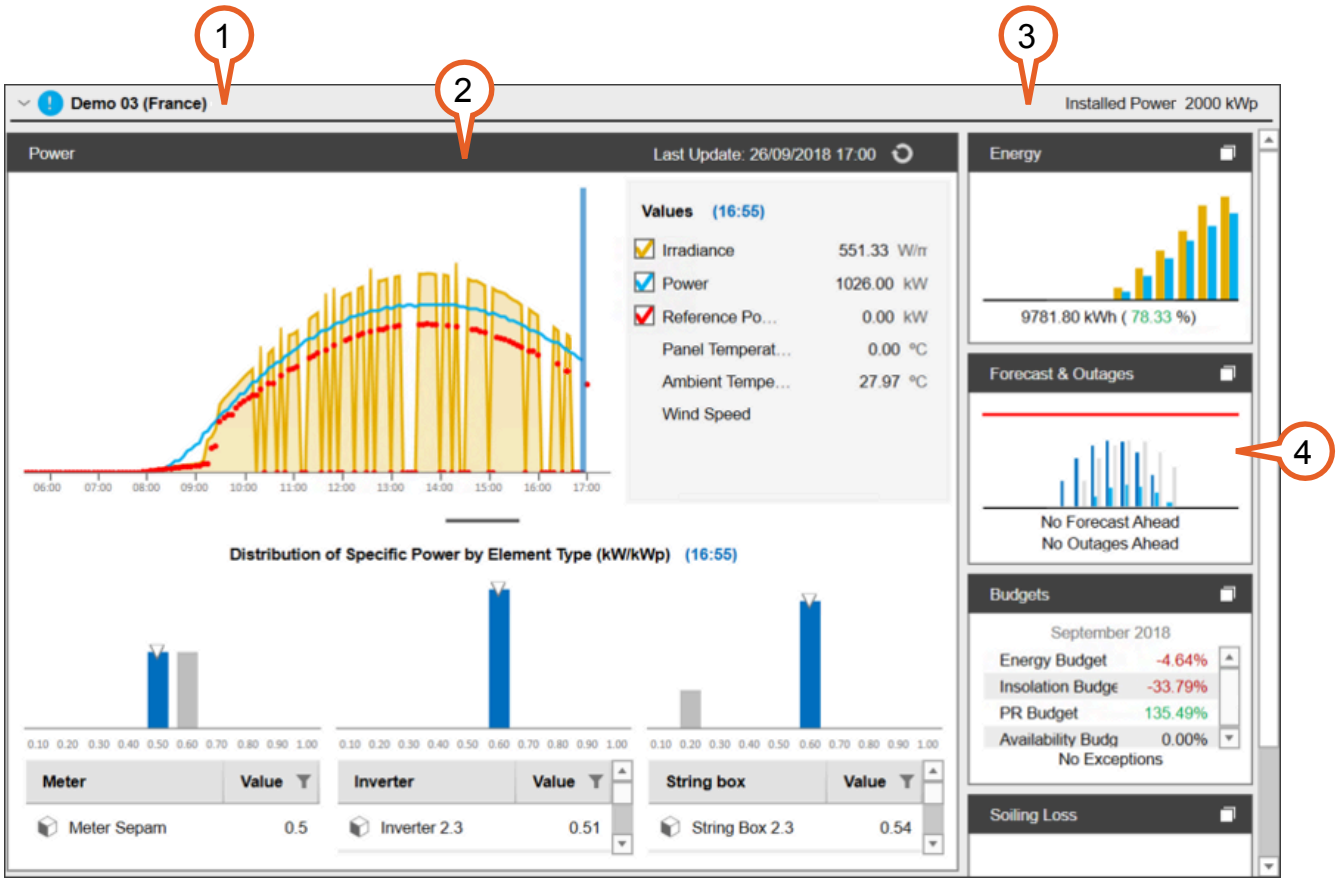
The Plant Dashboard module is the main tool to monitor and analyze your plant's performance with visual KPIs that compare multiple parameters.

The information on the Plant Dashboard module is organized into panels:

- Plant information
- Energy
- Power
- Forecast & outages
- Soiling loss
- Budgets

The main area displays the currently selected panel. The other panels are collapsed on the side, displaying condensed information. When you expand one of the panels on the side, the one currently displaying is automatically minimized.

Plant Dashboard module



1. Plant information: This panel is collapsed by default. When expanded, it displays

the plant name and the most relevant alarm for the plant (if any). Click the icon to expand the plant information, and click the icon to collapse it. For further information, see Plant Information Panel.

2. Main panel: The main panel area displays the KPIs and charts of the selected panel. You can refresh the data by clicking the icon on the top-right of the panel. The last data update is displayed on the top-right of the panel.
3. Installed power: displays the currently installed power of the plant.
4. Side panels: Minimized panels are displayed here and display a summary of the panel data.

   Click the icon on the top-right of a panel to expand it. For more information, see the sections corresponding to each panel:

   - Energy
   - Power
   - Forecast & Outages
   - Soiling Loss
   - Budgets

Budgets panel

The Budgets panel provides you with charts to monitor the productivity of your plant by comparing the actual production to the budgeted production.

Additionally, you can create exceptions to the availability and performance ratio calculations.

> ⓘ NOTE: You can display up to four charts on the Budgets panel. Contact to your GPM representative if you want to customize the charts on display.

Budgets panel

1. Date range: toggle between **Month** and **Year** to display monthly data or yearly data.

2. Date picker: select the month or year, according to the date range on display.
3. Data Granularity: toggle between **Months** and **Days** to display monthly or daily data.

   ⓘ NOTE: This toggle is only available for monthly data.

4. Charts: displays the budgeted data of the selected date range. You can select a specific date range by clicking a date on the chart or by dragging-and-dropping the blue date range picker ().

   Values with gray backgrounds represent exceptions.

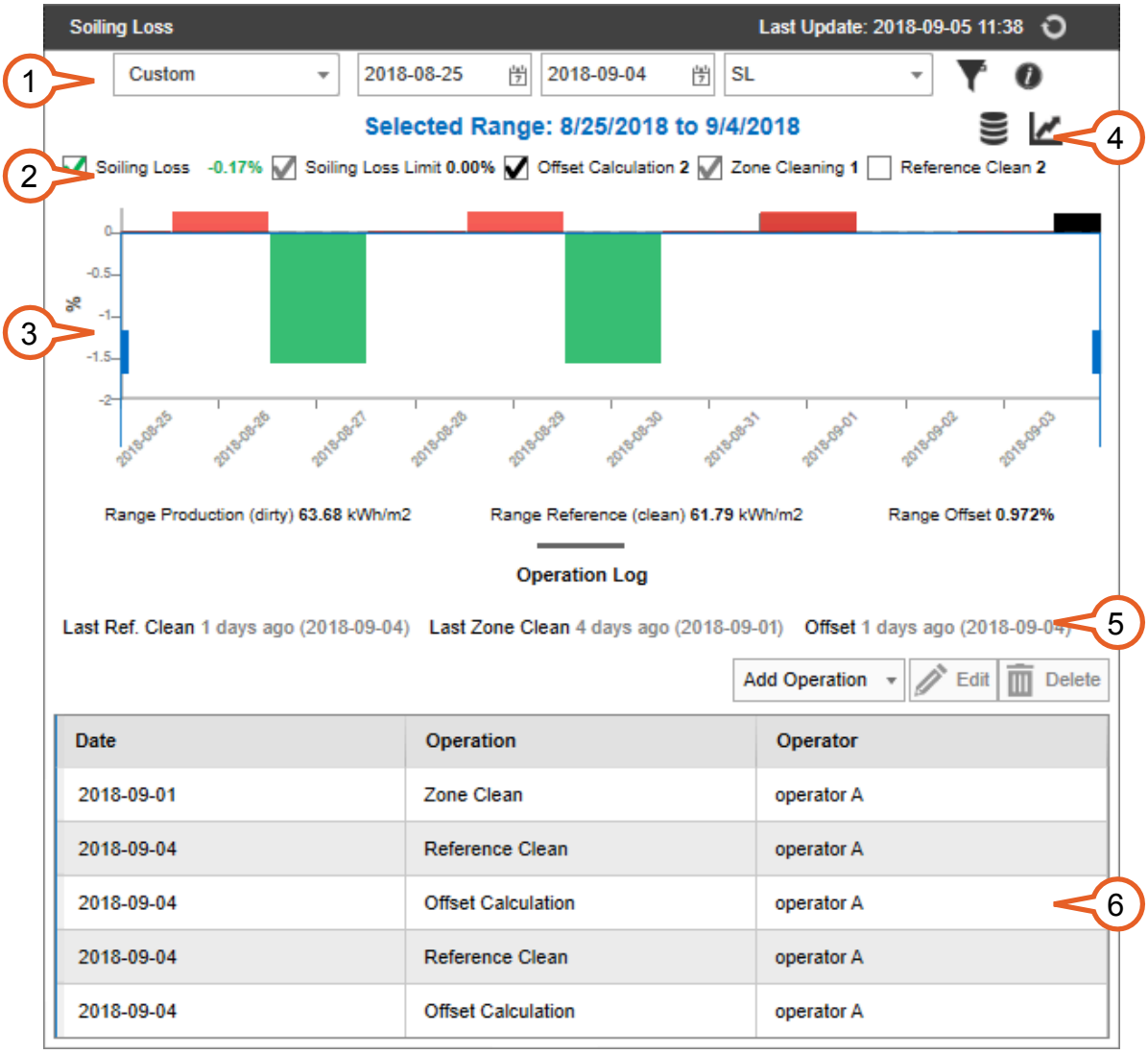
5. Exceptions list: displays the list of exceptions for the date range on the chart.

   Click the Exceptions button do add an exception to the calculations. For further information, see Create Exceptions.

Energy panel

The Energy panel provides you with a chart to monitor the daily energy generated by your plant. The data in the chart is divided by hour and can be customized by changing the date and the time interval or toggling the values on display.

Soiling Loss panel



1. Date Range and Zone Picker: Select a date range and a zone for the data. You must click the icon to display the selected date range on the chart.
2. Values: provides a text legend of the data displayed on the chart. You can click a parameter to toggle it on the chart.
3. Chart: displays the soiling loss data of the date range that you picked. You can select a specific date or date range by clicking an hour on the chart or by dragging-and-dropping the blue hour range picker ( ). When you select a specific date or

date range, their values are displayed in the Value Ribbon.

4. Quick actions: click to display the chart data in the Data Viewer module or click to display the chart data in the Linear Chart Viewer module.
5. Last operations: displays the date of the latest maintenance operations.
6. Operation list: displays a list of the maintenance operations. You can add, edit, or delete maintenance operations using the action buttons.

   Click any column header to sort the list by that column. For more information, see Soiling Loss Operations.

Forecast & Outages panel

The Forecast & Outages panel provides you with tools to plan and monitor the stop or decrease of performance in your portfolio due to an electric network request or maintenance. The data displayed is hourly and can be customized by toggling the parameters.

Power panel

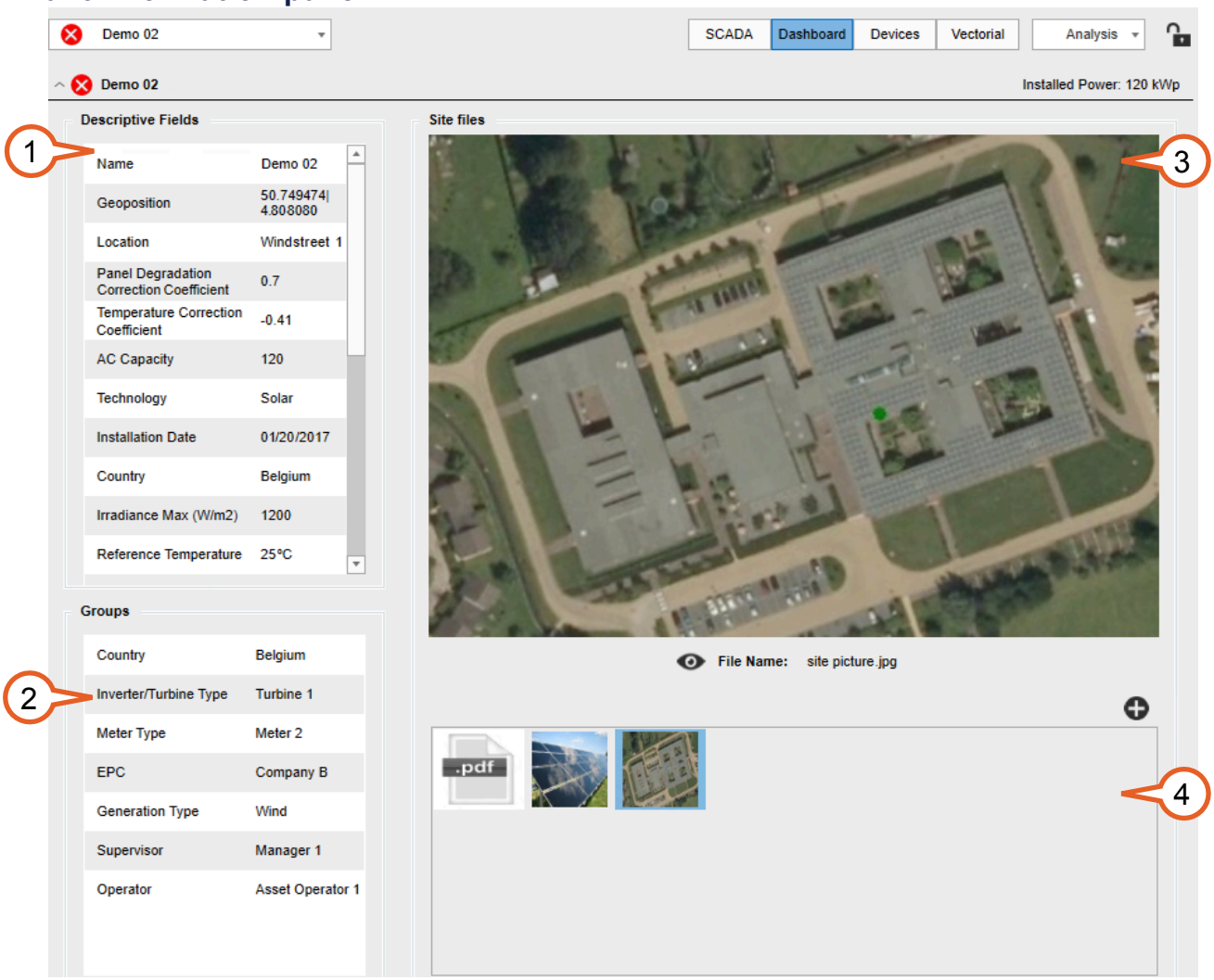
1. Chart: displays the power data of the date that you picked. By default, the entire hour range is selected. You can click a specific hour on the chart to display the hourly values in the Value Panel.
2. Values panel: provides a text legend of the data displayed on the chart. You can click a parameter to toggle it on the chart.
3. Distribution charts: display the device performance by grouping data in bins. Elements are charted comparing the actual performance to the declared peak performance during the time range selected on the main chart.

Click any bar on the chart to display the elements that belong to that bin in the table below.
Sort the table by the productivity Value by clicking on the header and filter the elements by clicking the icon. You can display any element on the Element Viewer by clicking the icon.

Plant Information panel

In the Plant Information panel, you can see general information about the selected plant and add plant-specific files. Click the ⌄ icon next to the plant name to expand or collapse the plant information.

Plant information panel



1. Plant description: general plant information is displayed here.
2. Groups: lists all the groups that the plant belongs to.

   Groups are a set of custom parameters used to classify the portfolio.
3. Plant image: displays an image of the plant. You can upload an image and select it from the plant files.
4. Plant files: upload and view plant-specific files.
   You can upload files by clicking the ⊕ icon and view uploaded files in a separate

window by clicking the  icon. The size limit for uploaded files is 2MBs.

Power panel

The Power panel provides you with a chart to monitor the daily power generated by your plant. The chart displays hourly data that can be customized by toggling the values on display.

Power panel

1. Chart: displays the power data of the date that you picked. By default, the entire hour range is selected. You can click a specific hour on the chart to display the hourly values in the Value Panel.
2. Values panel: provides a text legend of the data displayed on the chart. You can click a parameter to toggle it on the chart.
3. Distribution charts: display the device performance by grouping data in bins. Elements are charted comparing the actual performance to the declared peak performance during the time range selected on the main chart.

Click any bar on the chart to display the elements that belong to that bin in the table below.
Sort the table by the productivity Value by clicking on the header and filter the elements by clicking the icon. You can display any element on the Element Viewer by clicking the icon.

Soiling Loss panel

The Soiling Loss panel allows you to monitor and manage the loss of performance caused by soiling on solar panels. The chart displays daily data that you can customize by changing the date interval or toggling the values on display. You can also monitor data from multiple zones within the same plant.

The **Operation Log**, located at the bottom of the panel, allows you to monitor the latest maintenance operations and add new ones to the soiling loss calculations.

Power panel

1. Chart: displays the power data of the date that you picked. By default, the entire hour range is selected. You can click a specific hour on the chart to display the hourly values in the Value Panel.
2. Values panel: provides a text legend of the data displayed on the chart. You can click a parameter to toggle it on the chart.

3. Distribution charts: display the device performance by grouping data in bins. Elements are charted comparing the actual performance to the declared peak performance during the time range selected on the main chart.

   Click any bar on the chart to display the elements that belong to that bin in the table below.
   Sort the table by the productivity Value by clicking on the header and filter the elements by clicking the icon. You can display any element on the Element Viewer by clicking the icon.

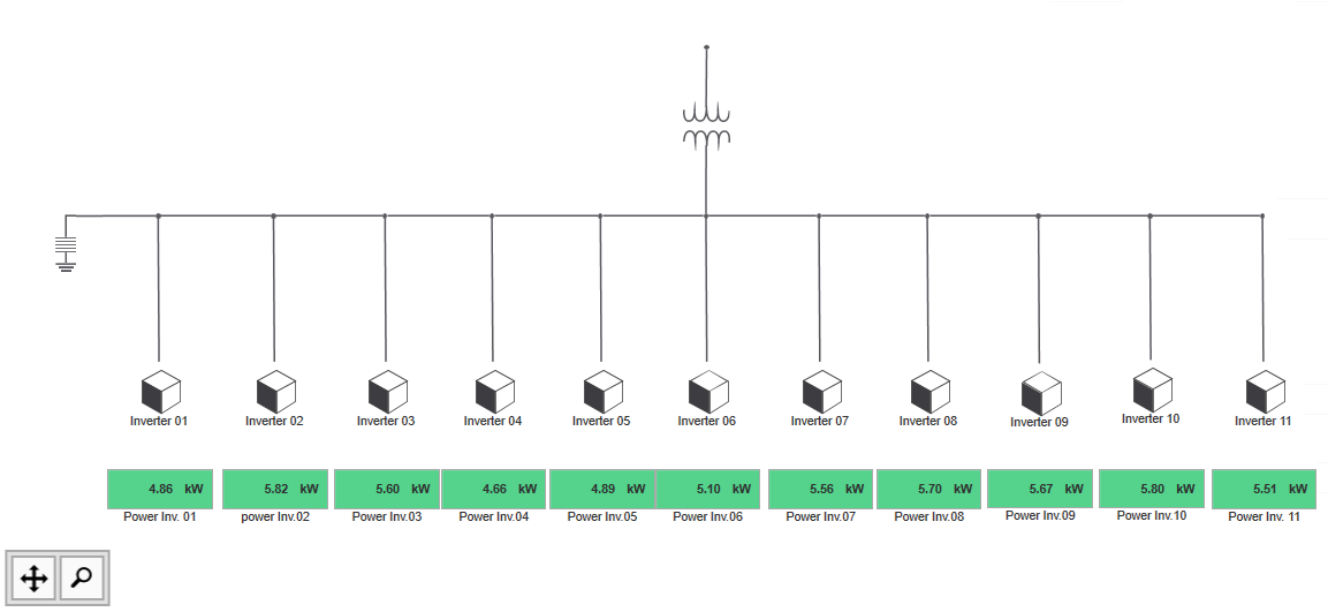
Vectorial Layout module

The Vectorial Layout module provides you with a multi-layered, custom vectorial schema to help you understand a plant's structure, its relationships, and the status of its elements. Within the Vectorial Layout, you can also send pre-defined commands to the hardware devices in your plant.

Access the module by clicking the  **Vectorial** tab in the upper area of the screen.

> ⓘ NOTE: To customize your Vectorial Layout, contact your GPM representative.

Vectorial layout



Navigate the Vectorial Layout

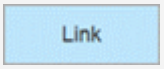
Action	Description
Mouse scroll	Use the scroll wheel on your mouse to zoom in and out of the Vectorial Layout. To zoom in, scroll up. To zoom out, scroll down.
Drag and drop	Click and hold on a point of the layout, then drag it anywhere to move around the interface.
Single click	Click on elements of the Vectorial Layout to interact with them. For further information, see the reference below.
Auto-fit	Click the icon at the bottom-left of the Vectorial Layout to automatically resize it and fit the screen.
Zoom	Click the icon at the bottom-left of the Vectorial Layout to set the zoom manually and preview the area being zoomed. Click the ✕ icon to close the zoom dialog.

Vectorial Layout elements

You can interact with some elements of the Vectorial Layout. This reference lists the elements with which you can interact.

> ⓘ NOTE: The background, element and icon colors in your vectorial layout are customizable. Contact your GPM representative if you want to customize them.

Element	Name	Description
	Alarm Counter	Counts the number of specific alarms for a specific set of devices. It is possible to set limits for the counter, so that its color changes from white to red when alarms exceed the configured number. When the alarms exceed the configured number and the counter turns red, you can click the box to display related alarms in the Alarms module.
	Command	Click to execute a command or a metacommand on one or more devices. If the command is configured to be executed on multiple devices, you can select the devices to which the command must be sent in the pop-up window. For more information, see Send a Command Sequence from the Vectorial Layout.
	Connection point	Displays the layout nodes.
	Data	Displays information that is retrieved from a parameter and is automatically refreshed at regular intervals. The box icon can be customized and up to three statuses can be set on every box. For example, the color can be set to change when a certain condition is met. Click a data box to display the values in the Live Viewer module.

	Element	Represents a physical device that is communicating with the application. If there is an active alarm on the element, an alarm icon is displayed next to it. Click an element to display further information in its relevant section.
	Icon	A static visual reference to a physical device in your plant. For further information, see Send a Command Sequence from the Vectorial Layout.
	Label	A text label used to identify elements.
	Line	Connects various physical or virtual elements in your setup. Lines can be assigned custom color-coding that changes when a specific condition is met.
	Link	Links the current layer to other layers in the layout. Click it to display the linked layer.
	Switch	Switches are elements that behave as a data box. This means that they can have values that change over time.

Vectorial icons

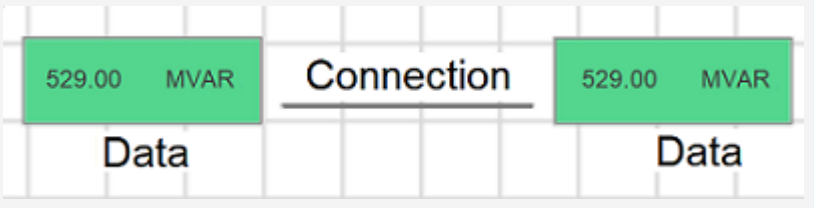
The application uses a default set of icons to identify specific device or element types in the Vectorial Layout.

Icon	Name
	Breaker
	Circuit breaker
	Current transformer
	Earth ground
	Fuse
	Generator
	Medium
	Meter
	Potential transformer
	Surge arrestor
	Transformer
	Transformer Type X
	Three-winding transformer

Configuring data objects and connection objects

You can configure data objects and connection objects using an XLSX template or the corresponding configuration window in the grid.

ⓘ NOTE: A **data object** is an element used to monitor a datasource value of a device, while a **connection object** is a line used to connect several objects.



Configuring data objects and connection objects using the XLSX template

I

- SourceType: Set the type of source the layer must use. You can enter one of the following options:
  - Datasource: It defines the type of source required in the *DatasourceId* column. When configuring a Monitored DS or Custom DS, you must provide a specific value for the *DatasourceId*. Additionally, you need to fill in the *ElementID* field in both cases.
  - ElementParameter: It indicates that the configuration of the object is determined by the *ElementParameterID* field. You must set a specific value for the element parameter during setup (parametrization).
- DataSourceComponentId

> ⓘ NOTE: This column can only be used in GPM Plus. Do not configure it as it will not display any data.

This is an example of how to configure layer settings for data and connection objects using the XSLX template:

Item	SourceType	ElementId	ElementParameterId	DatasourceId
DataElement PARAMETER	ElementParameter	247	76	0
DataElement MONITORED DATASOURCE	Datasource	247	76	0
DataElement CUSTOM DATASOURCE	Datasource	0		20941

Configuring data objects and connection objects in the grid

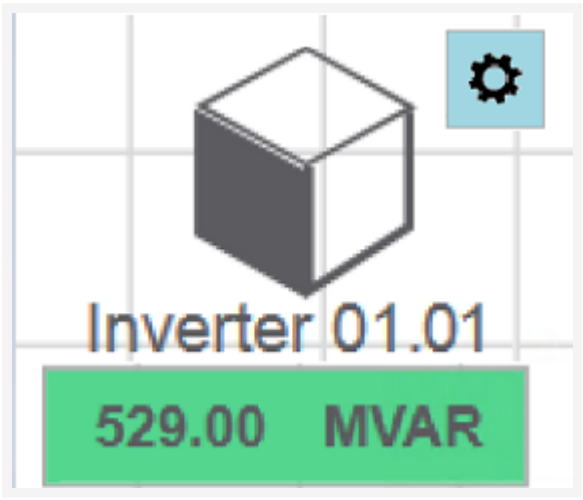
You can configure data objects and connection objects using the corresponding configuration window. You have to choose the desired element in the plant and assign to it parameters.

Configuring data objects in the grid

You can configure data objects using the corresponding configuration window.To configure data objects, follow these steps:

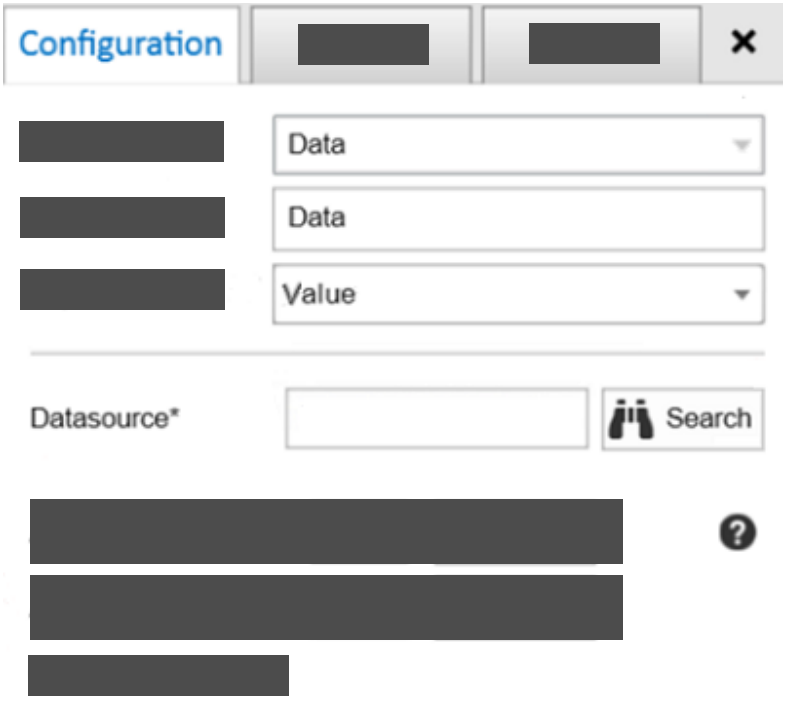
1. Click on the configuration button for the required data object.

Data object configuration



2. In the configuration screen, go to the *Datasource* field and click the Search button.

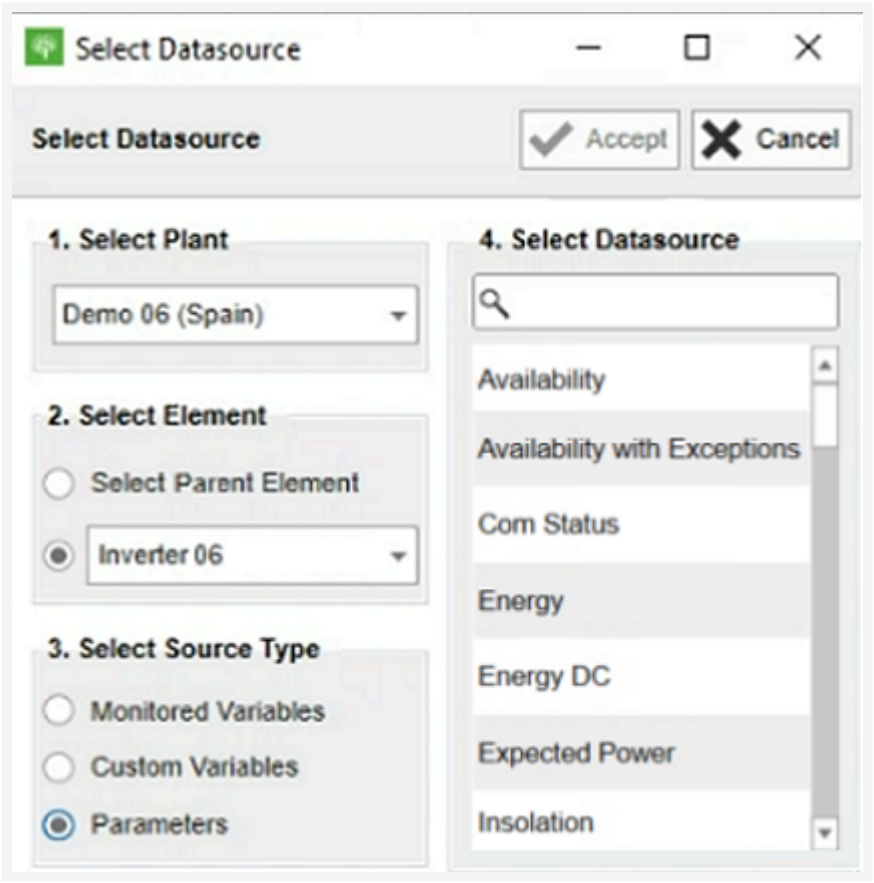
New Datasource field



3. In the Select Datasource window, do the following:

- 2. Select Element: select the element in the plant to which you want to assign parameters.
- 3. Select Source Type: select **Parameters**. The list of available parameters will then be displayed.
- 4. Select Datasource: choose the required parameter.

Select Datasource

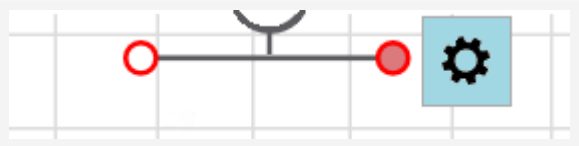


Configuring connection objects in the grid

You can configure connection objects using the corresponding configuration window. To configure connection objects, follow these steps:

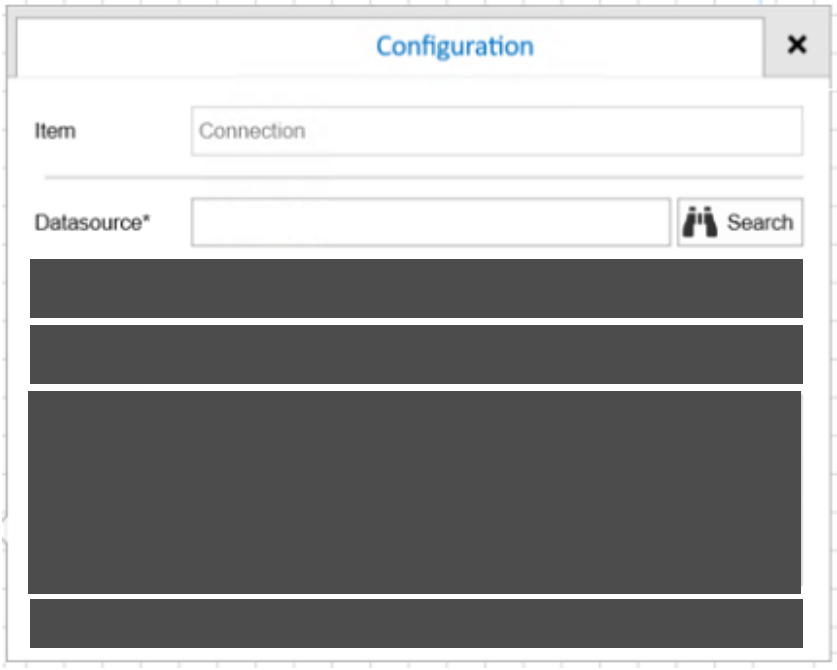
1. Click on the configuration button for the required connection object.

   New Datasource field

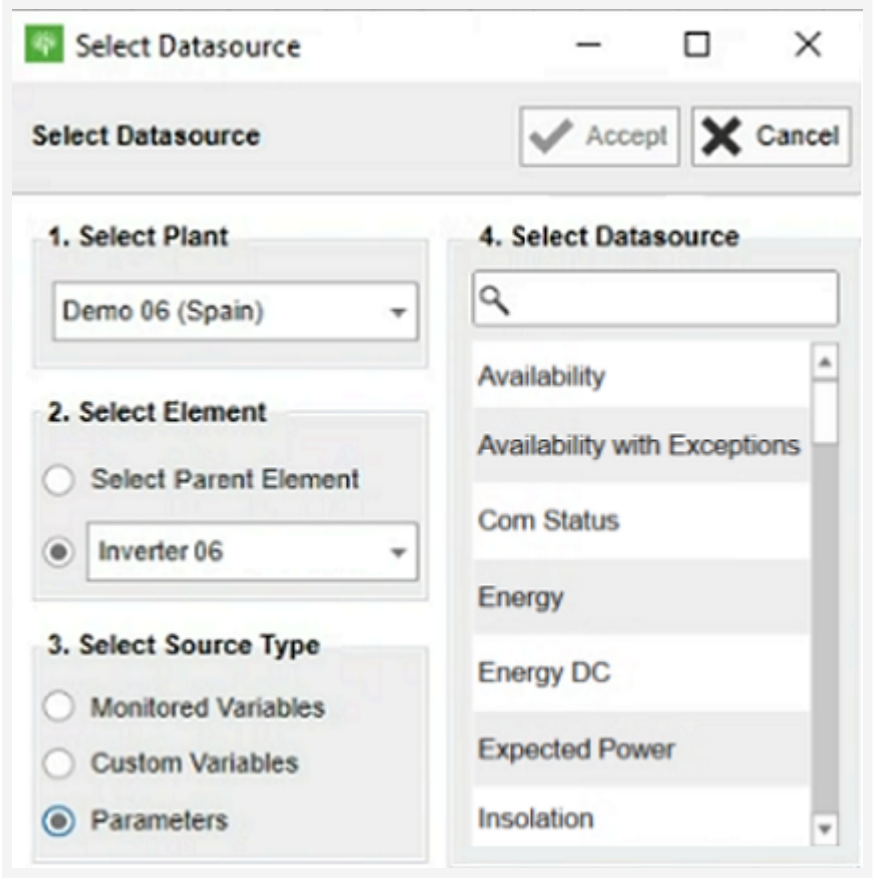
2. In the configuration screen, go to the *Datasource* field and click the Search button.

   New Datasource field

3. In the Select Datasource window, do the following:

Select Datasource



2. Select Element: select the element in the plant to which you want to assign parameters.

3. Select Source Type: select **Parameters**. The list of available parameters will then be displayed.

4. Select Datasource: choose the required parameter.

5

Result

SCADA layout module

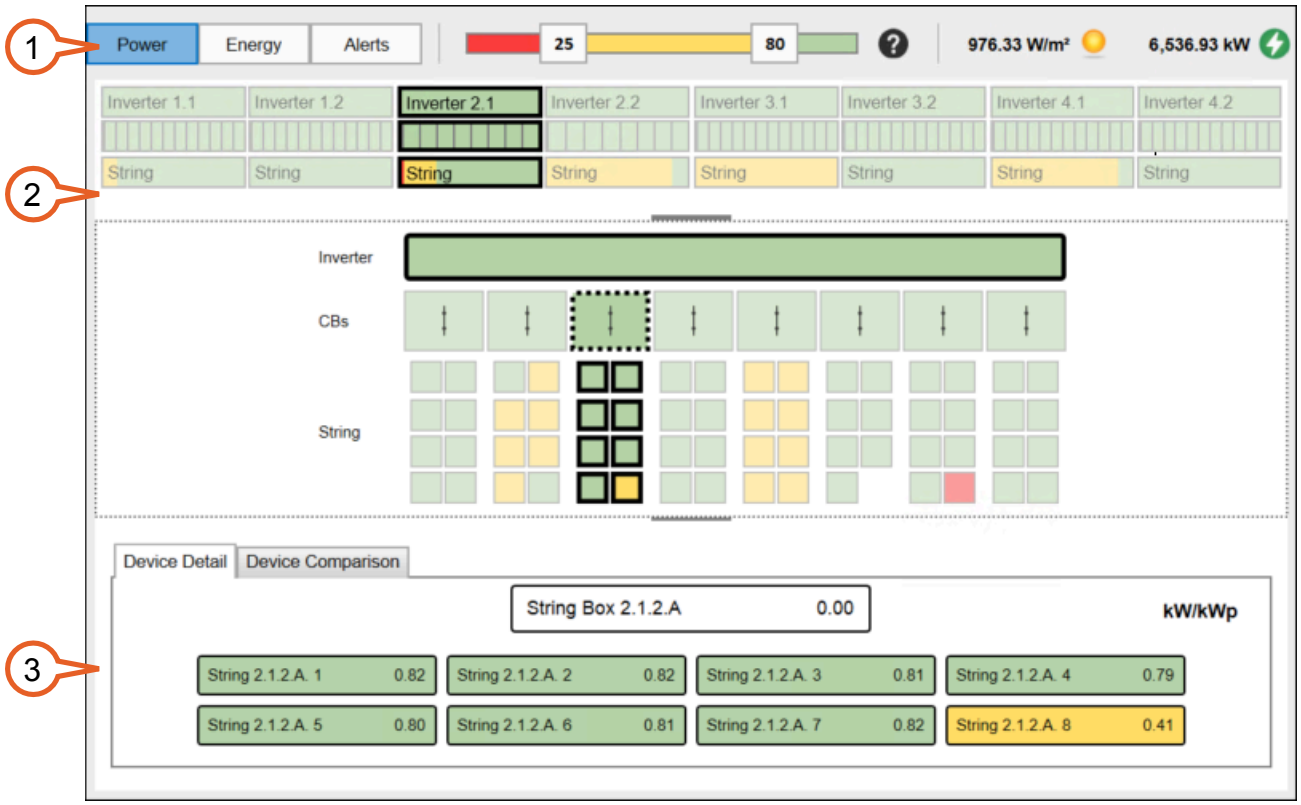
The Supervisory Control and Data Acquisition (SCADA) layout modules provides a global view of the plant efficiency and of the real-time status of the plant devices.

The SCADA Layout module uses color-coded cells to represent the performance of the devices. Each cell represents one element, and the cell size depends on the position of the element in the hierarchy. The performance of devices is calculated using the value of the best performing device as a reference and then expressed in percentage. The reference device can either be the best performing device of the same type or the best performing sibling device. The SCADA Layout also offers tools to analyze a single device and to compare multiple devices.

This module has three modes to analyze efficiency:

- **Power**: performance is calculated by comparing the device performance against a reference device for each device type. The reference is the latest and highest normalized power value for each device type.
- **Energy**: performance is calculated by comparing the device performance against a reference device for each device type. The reference is the highest daily normalized energy value for each device type.
- **Alarms**: each device is colored with the color of its most representative alarm that is active at the time. There is no comparison between devices in this mode.

SCADA layout



1. Main Menu
2. Element Hierarchy
3. Element Analysis

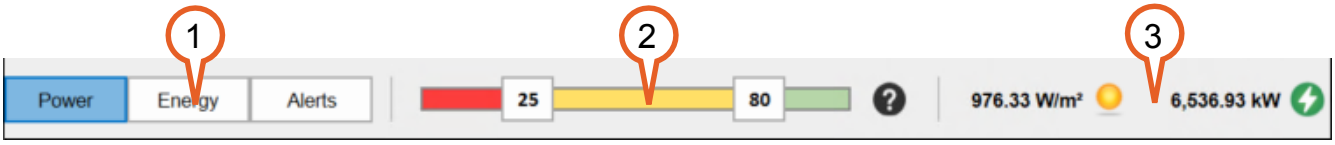
Color	Performance
Dark green	The device used as a reference to calculate the performance of other devices.
Green	The device is performing above the highest threshold.
Yellow	The device is performing between the lowest and highest thresholds.
Red	The device is performing below the lowest threshold.
White	The device is not communicating with the system and data cannot be retrieved.

Main Menu

The main menu is where you can change the SCADA Layout mode, customize the threshold for device comparison, see the active alarms, and see the plant weather and the latest total power or energy produced by the plant.

> ⓘ NOTE: The options available on the user interface vary depending on the mode that you select.

Main menu



1. Mode selector: click to switch between **Power**, **Energy** and **Alarms**.
2. Slider bar: click and drag the sliders to define the thresholds for performance. Click the ? icon to display the legend.

   > ⓘ NOTE: The slider bar is only available in Power and Energy modes.

   OR:

   Active alarms: displays a count of the active alarms, divided by type. Click the ? icon to display the legend.

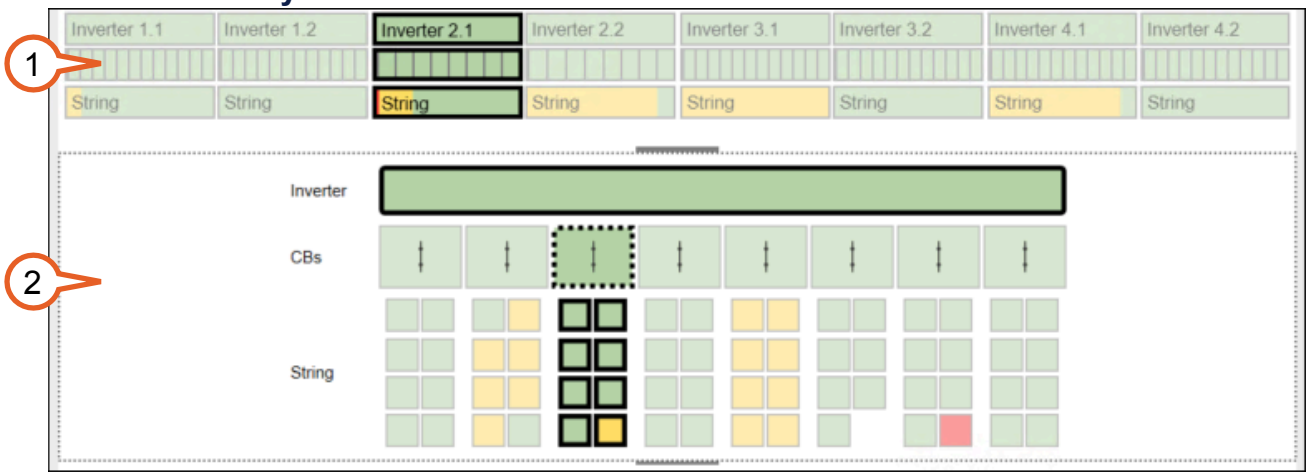
   > ⓘ NOTE: This is only available in Alarm mode.

3. Plant information: displays general information about the plant. The information depends on the selected mode:
   - **Power**: total plant irradiance, plant weather, and total plant power.
   - **Energy**: total plant insolation, plant weather, and total plant energy.
   - **Alarms**: available peak power percentage and the number of dataloggers currently communicating with the system.

Element Hierarchy

The Element Hierarchy is a visual representation of the plant production hierarchy with parent sets of elements on top, followed by child sets of elements. For example, inverters are on the top lines, followed by string boxes, which in turn are followed by string.

Element hierarchy



1. Plant units: displays the high-level hierarchy by units. For every unit, you can see the number of levels and use the color-coding to understand the general performance of each level. The last hierarchical level can be customized to condense all the information of its devices in a single cell. This cell is filled proportionally with the device performance color coding.

   Click a unit to display its devices in the Plant elements area.

   ⓘ BEST PRACTICE: We recommend activating the Plant Units area for plants with a high number of devices, when it is needed to divide the plant in smaller units.

2. Plant elements: displays the devices and their granular hierarchy. Click a device to display its details in the Device Details area or to add it to the Device Comparison areas. The selected device is highlighted with a dotted line and its parent and child devices with a full line.

   ⓘ BEST PRACTICE: Switches cannot be compared and have only two possible statuses:

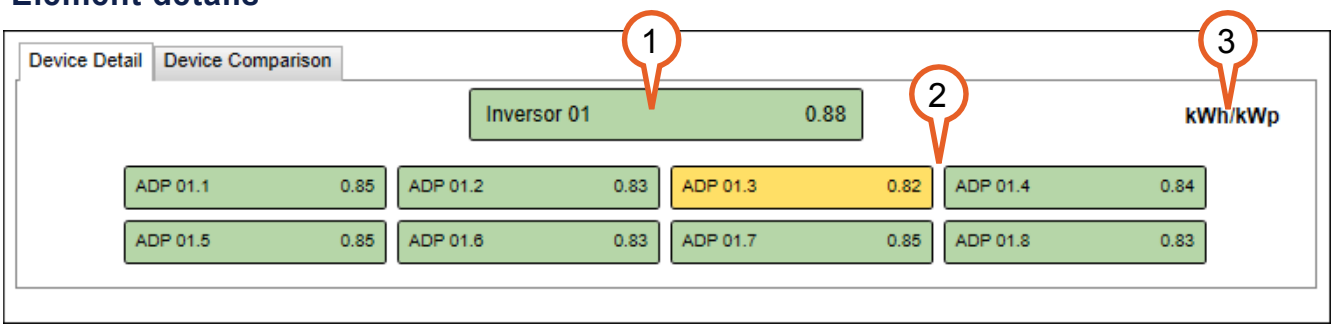
   - The switch is open.
   - The switch is closed.

Element Analysis

The Device Analysis area consists of two tabs where you can see the device details and compare devices between them:

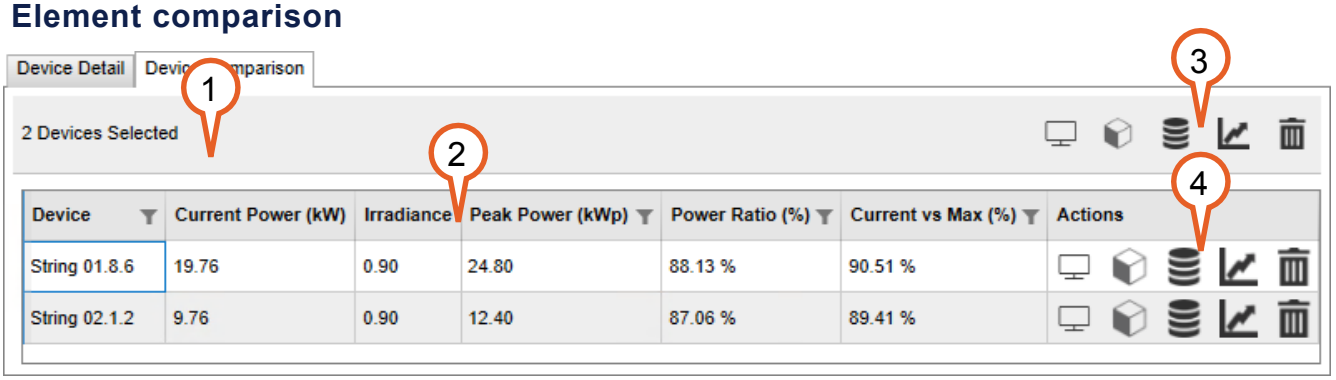
- Element details
- Element comparison

Element details



1. Selected element: displays the selected element as a cell. Its efficiency is expressed by the color-coding and by the value on the right side of the cell.
2. Child elements: the first level of child elements for the selected element are displayed as cells. Their efficiency is represented by the color-coding and by the value on the right side of the cell.
3. Performance unit: the unit used to express the element performance.

Element comparison



1. Element count: displays the number of selected elements.
2. Comparison table: displays selected elements. You can click any header to sort the table by that column.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon on any column

header for advanced filtering. For further information, see Advanced Filtering.

3. Group actions: click to open and compare the selected elements in other modules:
   - **Live Viewer**
   - **Element Viewer**
   - **Data Viewer**
   - **Linear Chart Viewer**

   Click the icon to delete all elements from the table.
4. Element actions: click to open the element in other modules:
   - **Live Viewer**
   - **Element Viewer**
   - **Data Viewer**
   - **Linear Chart Viewer**

   Click the icon to delete all elements from the table.

Plant elements module

The Plant Elements module helps users to quickly compare a group of devices and analyze their performance in real time or at specific moments in time. You can select the data to display from a pre-configured set of tables.

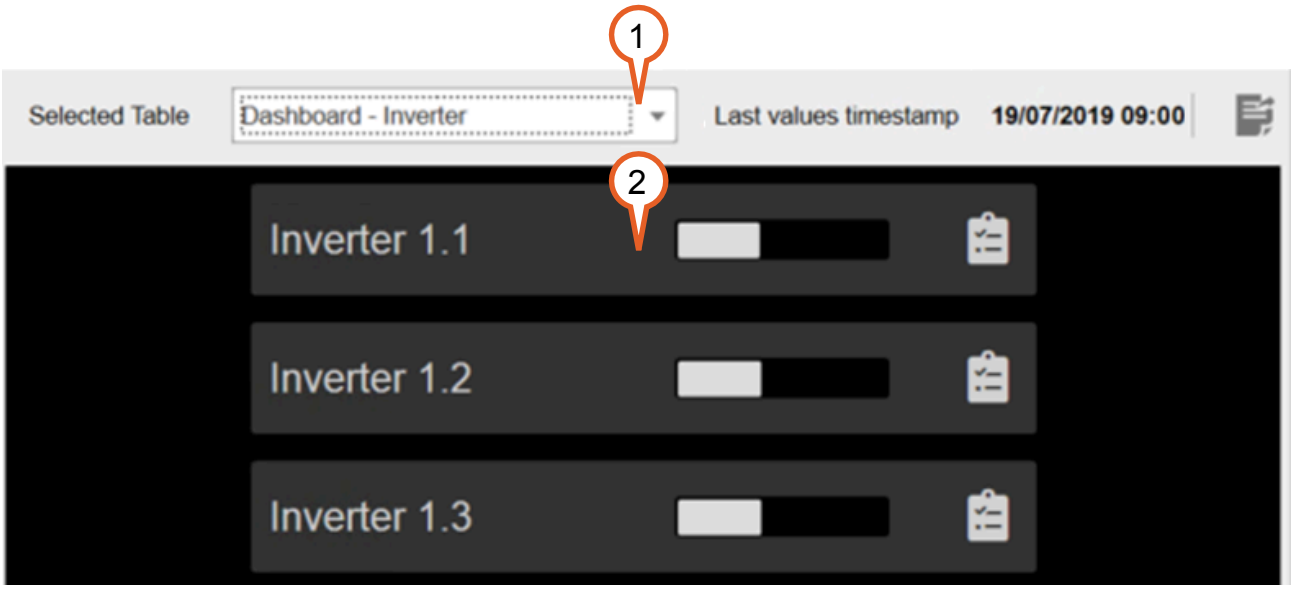
The Plant Elements module is divided in two sections:

- Upper bar: contains tools to customize the data on display.

  > ⓘ NOTE: The available options change based on the table that you select.

- Plant elements: displays the selected table using three pre-configured display modes: **Dashboard**, **Table**, and **Real-Time**.

Plant Elements module



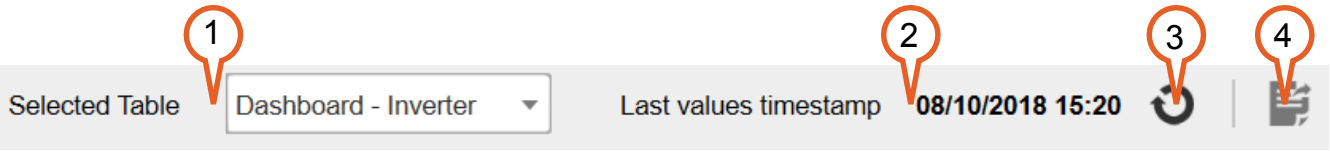
1. Upper bar
2. Plant elements

Upper bar

Use the Upper Bar to select and customize the data on display. The options available change based on the table that you select:

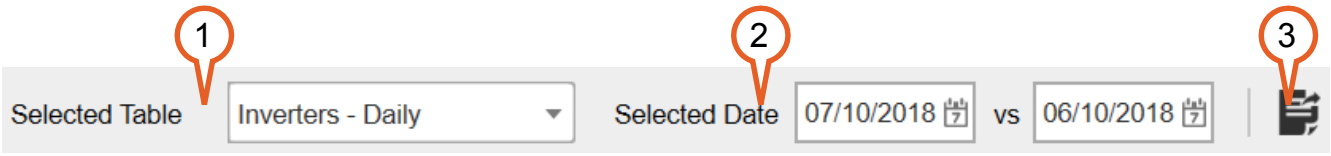
- Dashboard
- Comparison
- Real-time

Dashboard mode



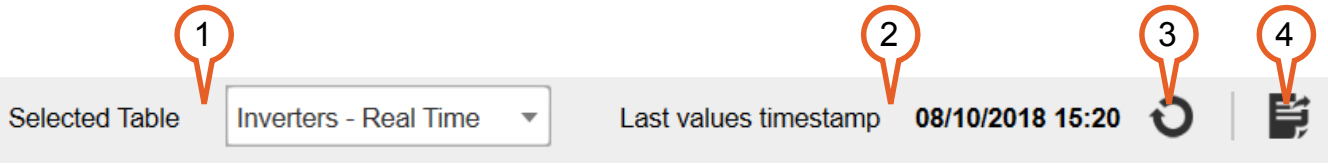
1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. Timestamp: displays the time for the value updates. Values are automatically updated at recurring intervals.
3. **Refresh**: click to manually update values.
4. Export: this button isdisabled in Dashboard mode.

Comparison mode



1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. **Comparison dates**: select the dates that you want to compare. The second date is the reference against which the values of the first date are compared.
3. **Export**: click to export the table to a Microsoft Excel format. For further information, see Export Data to File.

Real-time mode



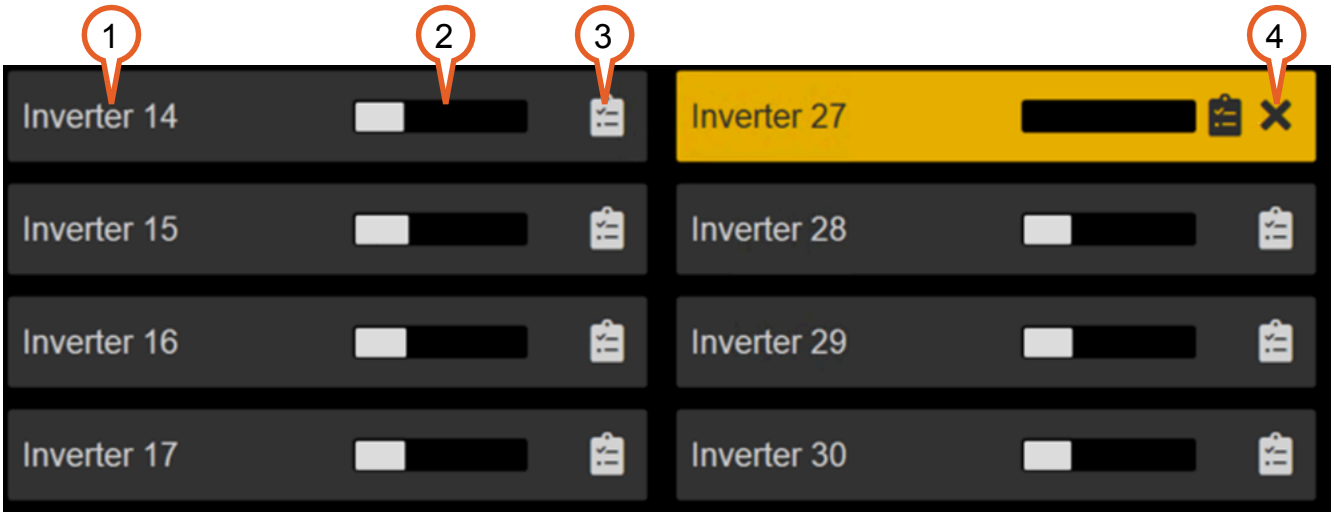
1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. Timestamp: displays the time for the value updates. Values are automatically updated at recurring intervals.
3. **Refresh**: click to manually update values.
4. **Export**: click to export the table to a Microsoft Excel format. For further information, see Export Data to File.

Plant elements

Monitor and compare devices that are at the same hierarchical level in a plant. This section has two display modes that change based on the table that you select on the upper bar:

- **Dashboard**: displays elements as cards.
- **Table**: displays the KPIs as a table.
- **Real-time**: displays the KPIs as a table.

Dashboard

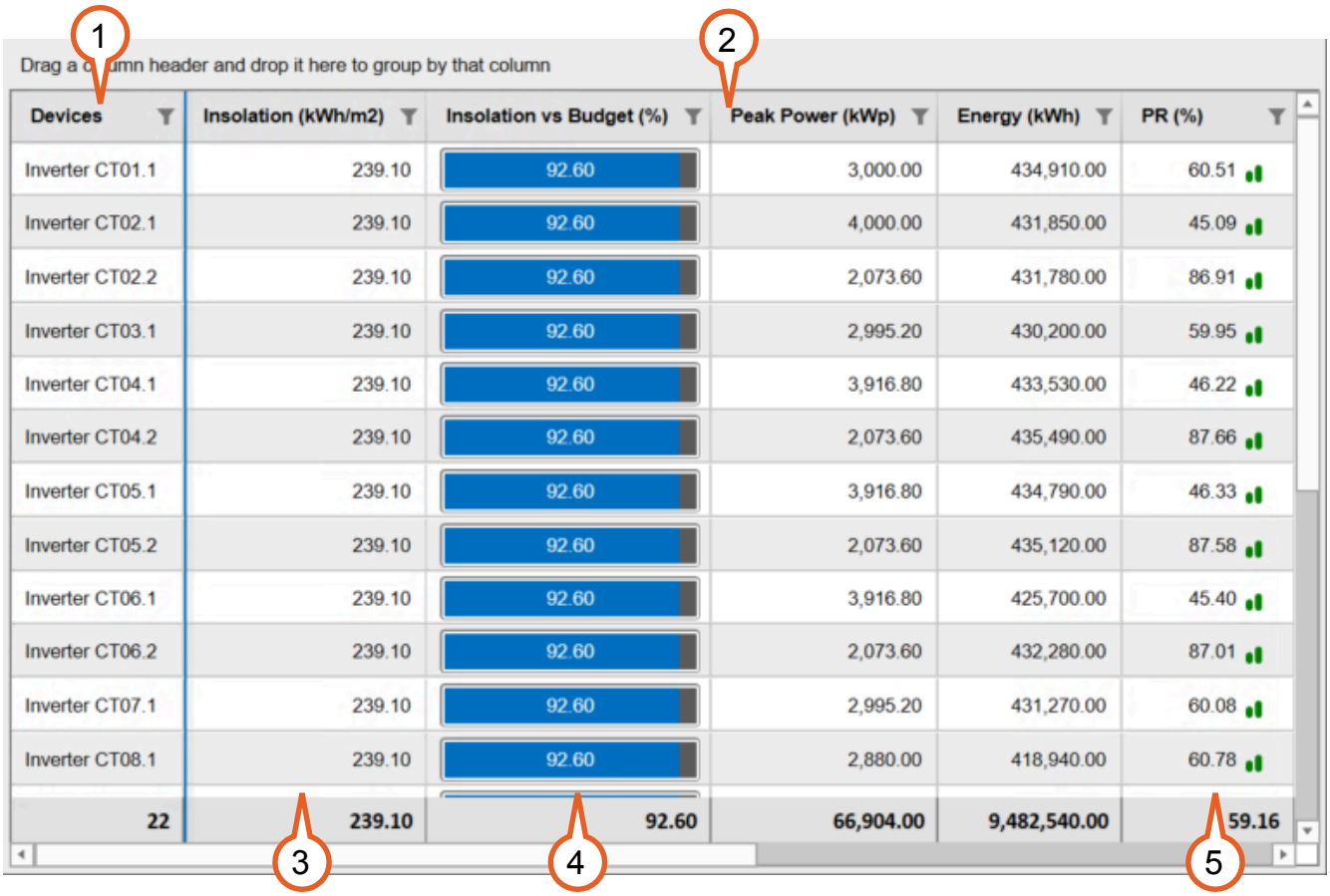


1. **Element name**: displays the name of the element. Click the name to display more details on the Element Viewer.
2. Status bar: displays a quick view of a particular KPI for the element.

   The KPI is calculated by comparing a reference value with the actual value received from the element. You can place your cursor on the bar to display the KPI details and click the bar to display the data in the Linear Chart Viewer module.

   The KPI on display is defined during the initial configuration of GPM SCADA. To define or customize KPIs, contact your GPM representative.
3. **Ticket status**: displays the Ticket icon when there is any active ticket related to the element. Click the icon to open the Edit Work Order dialog.
4. **Alarm**: displays the icon of the most relevant alarm for the element, if any. Place your cursor on the alarm to display the alarm message and click the icon to display further information about the alarm in the Alarm Information panel.

Table & Real-time



Data Viewer module or in the Linear Chart Viewer module.

1. **Elements column**: click a element icon to display its information on the Element Viewer. Right click a element to display the context menu and access additional options:
   - **View Element**: this functionality is inactive.
   - **Filter Element Alarms**: display the related alarms in the Alarms module.
   - **Alarm Information**: open the Alarm Information dialog for the active alarm.
   - **Element Viewer**: display the selected element on the Element Viewer.
2. **Header rows**: Click a column header to sort the table by the values of that column. You can rearrange columns by dragging and dropping the headers. This also groups elements by the selected parameter. Click the ✕ icon to reset the table grouping.
   Click the icon on any column header for advanced filtering. For more information, see Advanced filters.

3. **Values column**: display data as text.
4. **Status bar column**: displays data in a colored status bar that indicates the performance in percentage. The percentage is calculated by comparing the current KPI against a reference KPI.
5. **Health icon column**: displays colored icons that indicate the health of the element. The health is calculated by comparing the current KPI and a reference KPI. For more information about how to interpret the icons, see the Ranking Module section.

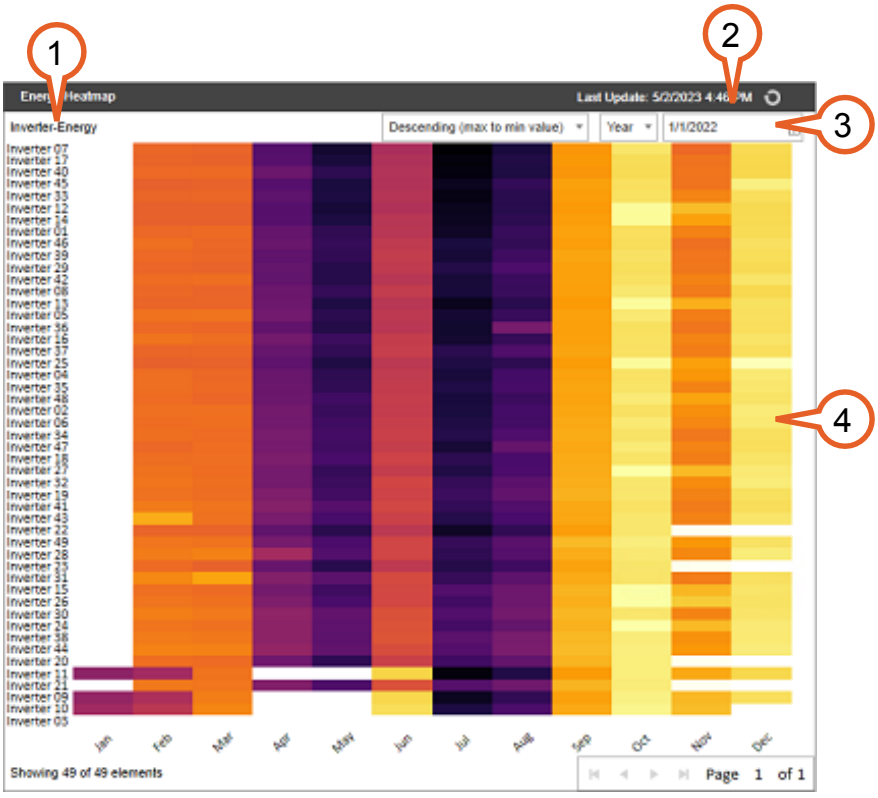
Heatmap

The Heatmap displays historical aggregated data at the element level (for example, inverters or wind turbines). This feature leverages GPM's Advanced Analytics to allow you to identify where and when assets in your portfolio are under-performing, allowing you to evaluate and address inefficiencies at the level of individual elements.

The default data available on the Heatmap are availablity, energy, and production ratio (PR). Each data appears as a separate Heatmap module in the Plant dashboard. You can sort the data alphabetically by element name, as well as by value, in ascending or descending order.

It is possible to customize the color-code for the percentage ranges, as well as to configure Heatmaps for other data. For more information on custom configurations, contact your GPM representative.

Heatmap



1. Element and data types on display.
2. Timestamp: informs you of when the data on display was retrieved. Click the ↻ icon to refresh and load the latest available data.
3. Display options:

- **Sorting**: open the drop-down menu to select how to arrange the data on the map:
  - **Alphabetical**
  - **Ascending (minimum to maximum values)**
  - **Descending (maximum to minimum values)**
- **Time span**: select the periods covered by the chart.
  - **Month**
  - **Quarter**
  - **Year**
- **Date selector**: open the drop-down menu to select the dates for the time period.

4. **Element performance**: hover over a cell to view a detailed information panel about the element's performance at a particular point in time:

Plant elements module

The Plant Elements module helps users to quickly compare a group of devices and analyze their performance in real time or at specific moments in time. You can select the data to display from a pre-configured set of tables.

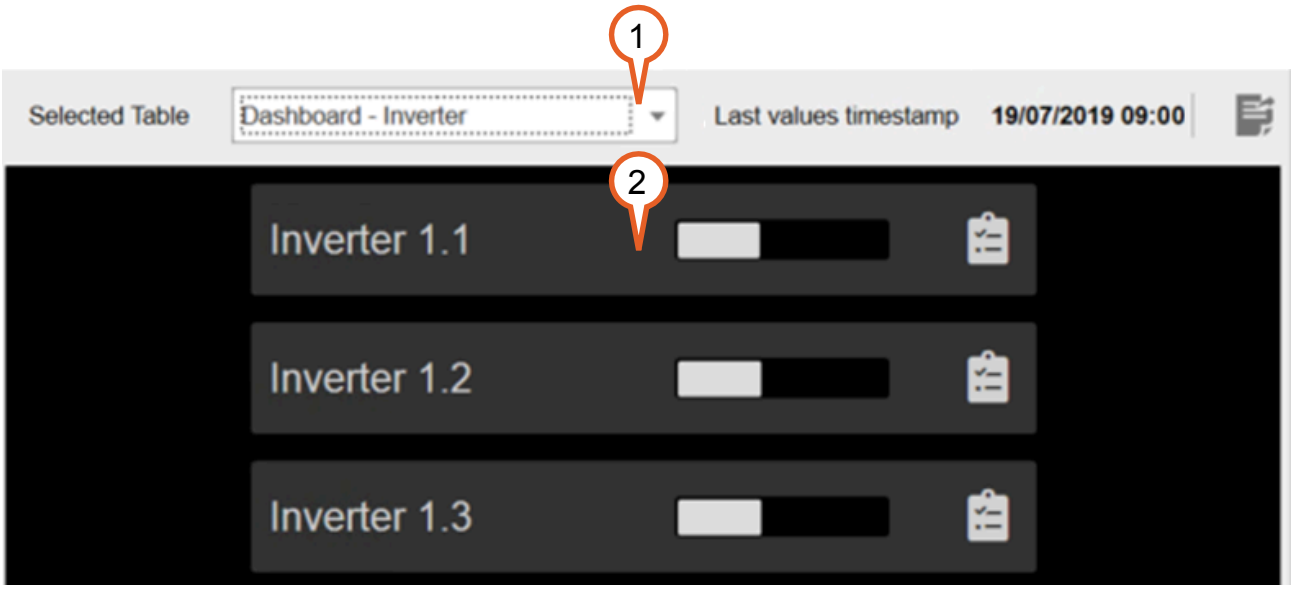
The Plant Elements module is divided in two sections:

- Upper bar: contains tools to customize the data on display.

  > ⓘ NOTE: The available options change based on the table that you select.

- Plant elements: displays the selected table using three pre-configured display modes: **Dashboard**, **Table**, and **Real-Time**.

Plant Elements module



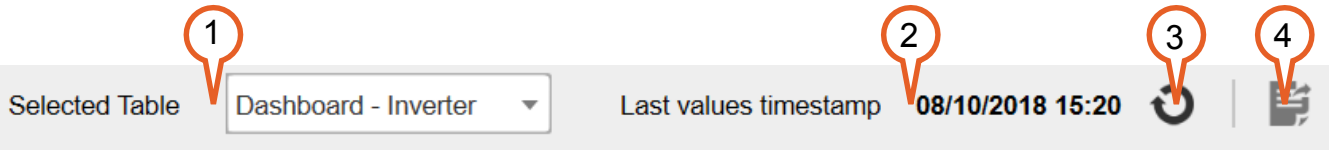
1. Upper bar
2. Plant elements

Upper bar

Use the Upper Bar to select and customize the data on display. The options available change based on the table that you select:

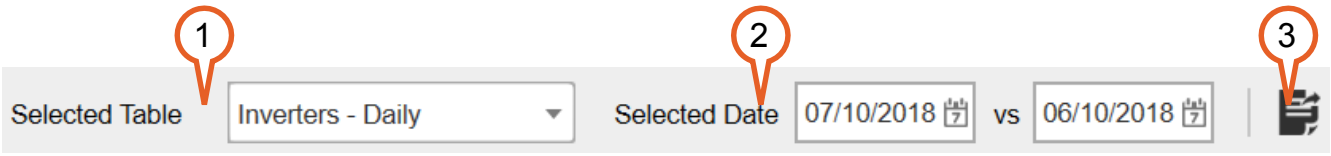
- Dashboard
- Comparison
- Real-time

Dashboard mode



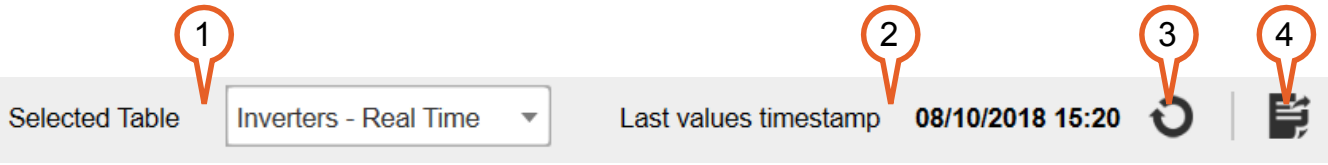
1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. Timestamp: displays the time for the value updates. Values are automatically updated at recurring intervals.
3. **Refresh**: click to manually update values.
4. Export: this button isdisabled in Dashboard mode.

Comparison mode



1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. **Comparison dates**: select the dates that you want to compare. The second date is the reference against which the values of the first date are compared.
3. **Export**: click to export the table to a Microsoft Excel format. For further information, see Export Data to File.

Real-time mode



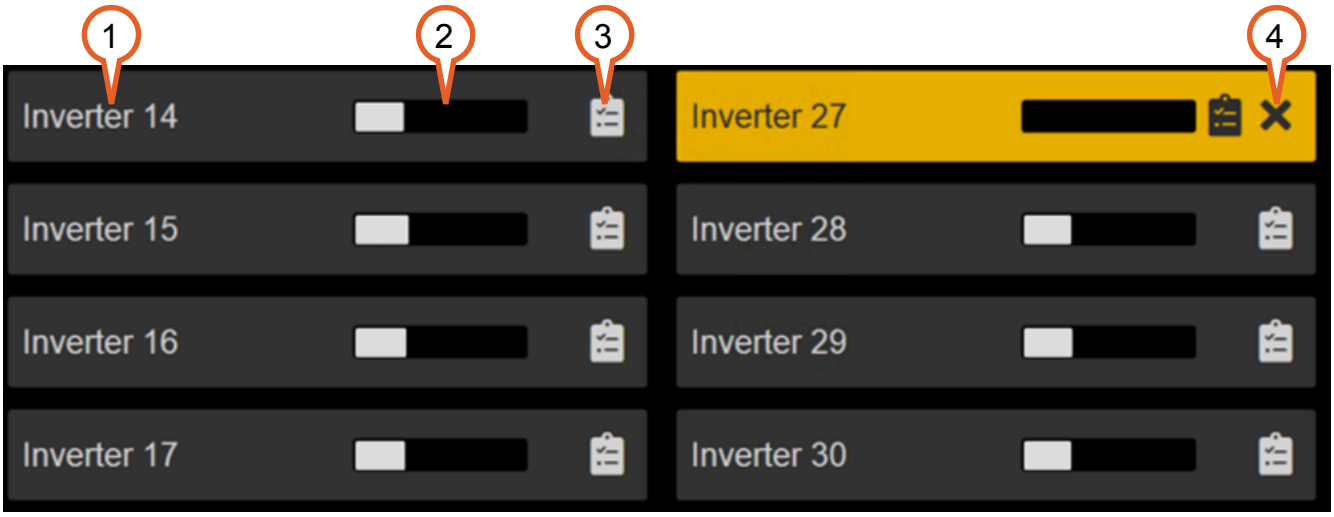
1. **Tables**: click to select a table from the drop-down list to display its data in the Plant Elements area.
2. Timestamp: displays the time for the value updates. Values are automatically updated at recurring intervals.
3. **Refresh**: click to manually update values.
4. **Export**: click to export the table to a Microsoft Excel format. For further information, see Export Data to File.

Plant elements

Monitor and compare devices that are at the same hierarchical level in a plant. This section has two display modes that change based on the table that you select on the upper bar:

- **Dashboard**: displays elements as cards.
- **Table**: displays the KPIs as a table.
- **Real-time**: displays the KPIs as a table.

Dashboard

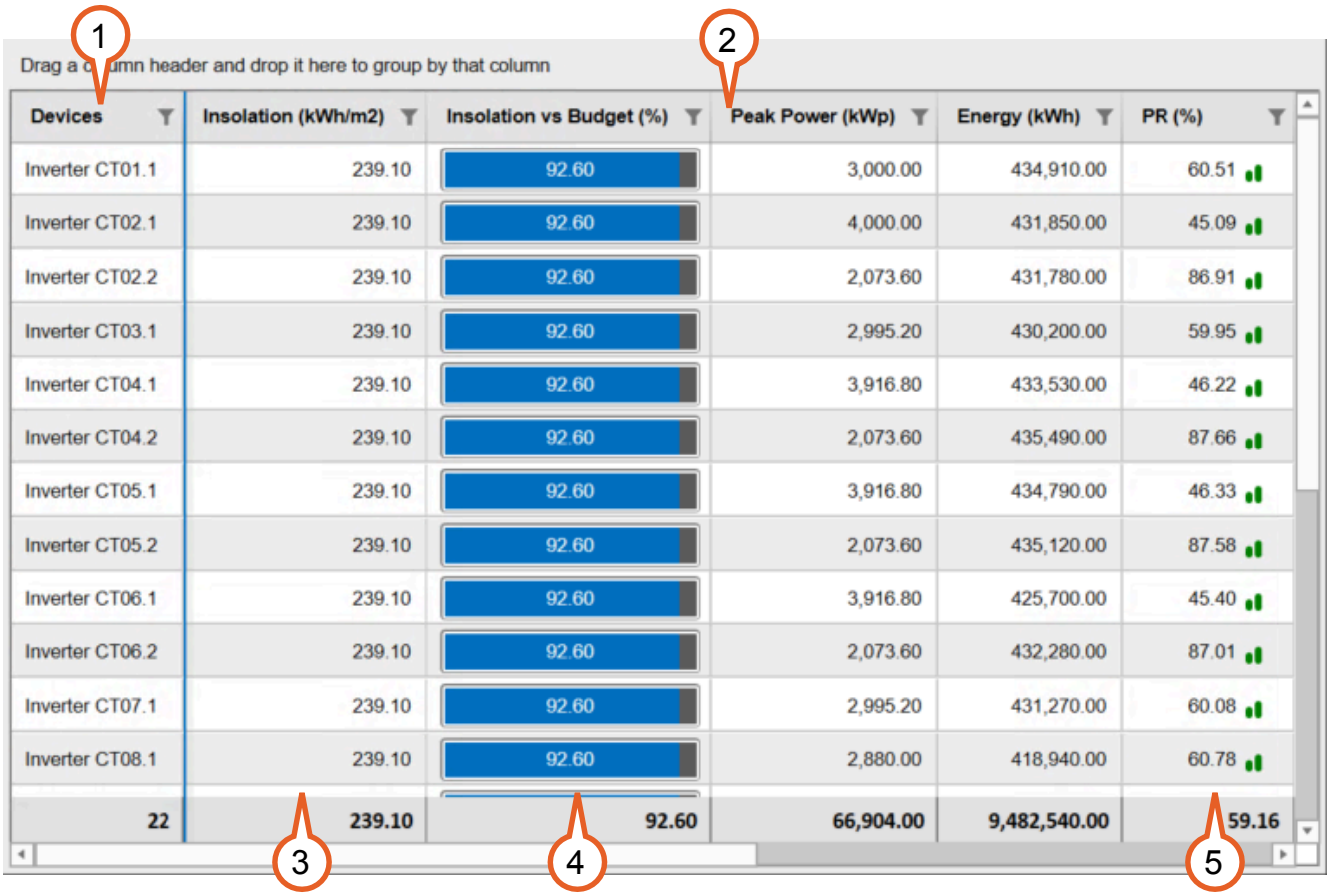


1. **Element name**: displays the name of the element. Click the name to display more details on the Element Viewer.
2. Status bar: displays a quick view of a particular KPI for the element.

   The KPI is calculated by comparing a reference value with the actual value received from the element. You can place your cursor on the bar to display the KPI details and click the bar to display the data in the Linear Chart Viewer module.

   The KPI on display is defined during the initial configuration of GPM SCADA. To define or customize KPIs, contact your GPM representative.
3. **Ticket status**: displays the Ticket icon when there is any active ticket related to the element. Click the icon to open the Edit Work Order dialog.
4. **Alarm**: displays the icon of the most relevant alarm for the element, if any. Place your cursor on the alarm to display the alarm message and click the icon to display further information about the alarm in the Alarm Information panel.

Table & Real-time



Data Viewer module or in the Linear Chart Viewer module.

1. **Elements column**: click a element icon to display its information on the Element Viewer. Right click a element to display the context menu and access additional options:
   - **View Element**: this functionality is inactive.
   - **Filter Element Alarms**: display the related alarms in the Alarms module.
   - **Alarm Information**: open the Alarm Information dialog for the active alarm.
   - **Element Viewer**: display the selected element on the Element Viewer.
2. **Header rows**: Click a column header to sort the table by the values of that column. You can rearrange columns by dragging and dropping the headers. This also groups elements by the selected parameter. Click the ✕ icon to reset the table grouping.
   Click the ▼ icon on any column header for advanced filtering. For more information, see Advanced filters.

3. **Values column**: display data as text.
4. **Status bar column**: displays data in a colored status bar that indicates the performance in percentage. The percentage is calculated by comparing the current KPI against a reference KPI.
5. **Health icon column**: displays colored icons that indicate the health of the element. The health is calculated by comparing the current KPI and a reference KPI. For more information about how to interpret the icons, see the Ranking Module section.

Power Plant Control

The Power Plant Control (PPC) allows you to control your plant by sending commands to its physical devices.

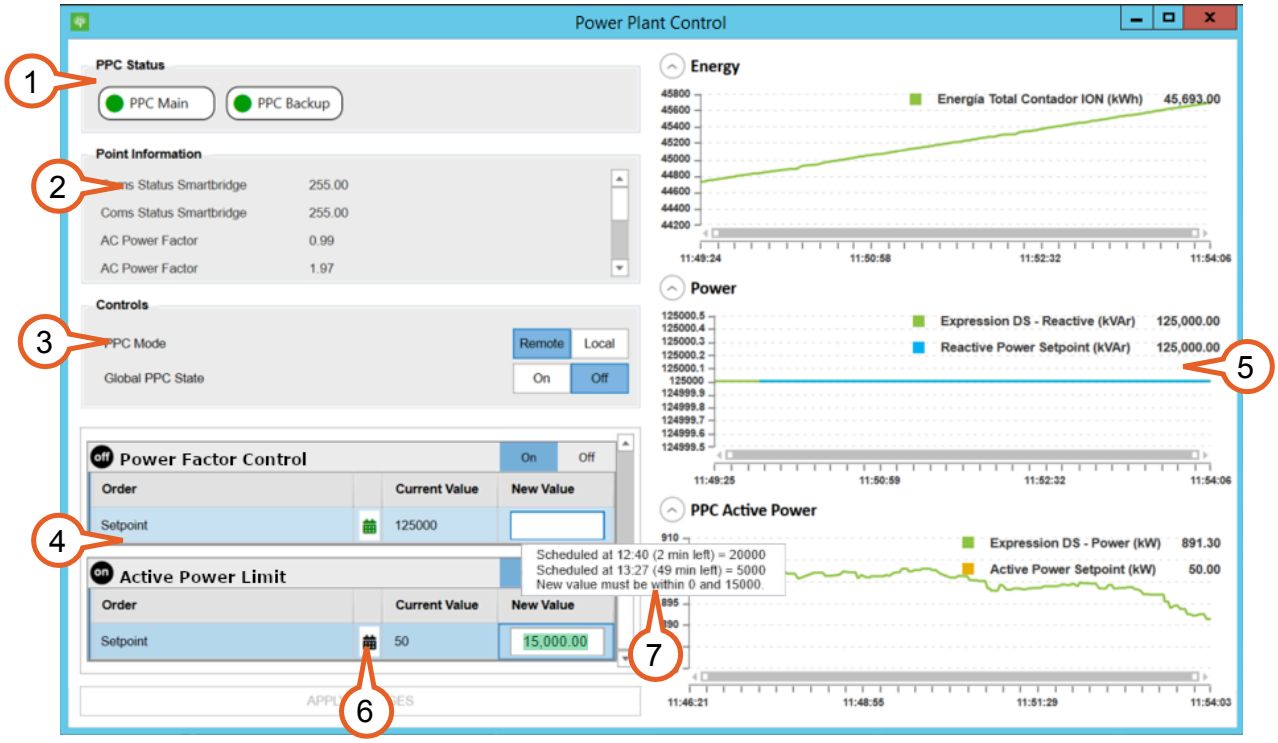
The PPC module also contains tools to monitor your plant's performance without leaving the interface, providing you with all the necessary tools to take quick decisions and act accordingly.

⚠ CAUTION: Use extreme caution and follow all the safety procedures before performing any action from the PPC module. These actions directly impact the plant.

⚠ CAUTION: For security reasons, actions taken in the PPC module are protected by a password and stored in the application log.

To access the PPC, click the icon on the Upper Bar.

PPC module



1. PPC Status: displays the communication status of the PPCs installed in the plant.
2. Point information: displays a pre-configured set of parameters that allow you to monitor your plant's performance.
3. **Quick controls**: click to change the PPC Mode and the Global PPC Status.

4. **Set point controls**: control your plant performance by sending Set Point values to your PPCs.

   Click **Apply Changes** to send the values to your PPCs.
5. Live charts: monitor your plant's performance in real time to see the effect of the Set Points values that you send to the PPCs.
   You can drag and drop on a chart to zoom in and collapse and expand graphs using the ⌄ and ⌃ icons.
6.  icon: displays the status of the scheduled command. If you click it, you can access the **Scheduler** dialog, where you can view and edit the scheduled command.
7. Active schedules: if you hover over the *New Value* input box, a tooltip containing information about the active schedules appears.

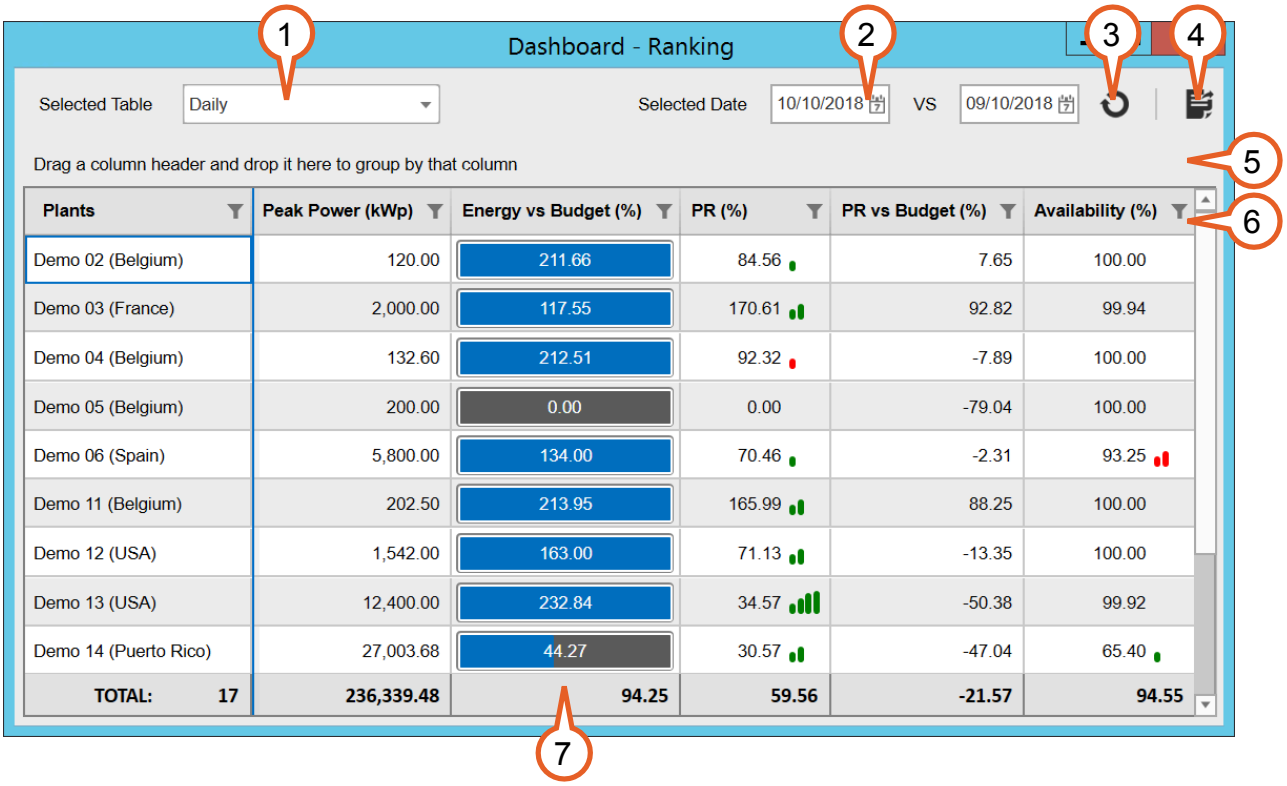
ⓘ NOTE: For more information on tasks related to set point controls, scheduled commands and command sequences, see the respective links.

Ranking

The Ranking module is a tool that allows users performing asset management tasks to analyze the status of their portfolio by comparing, grouping, and ranking the plants by their performance. The performance is calculated by comparing KPIs between two periods.

To access the Ranking module, click the icon in the Upper Bar and then click the icon.

Ranking Module



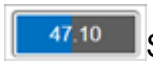
1. **Tables list**: select a table from the drop-down list. The available options change based on the table you select:
   - **Daily**: rank the KPIs of two different days.
   - **Monthly**: rank the KPIs of two different months.
   - **Real Time:** rank KPIs as they are retrieved in real time by the system. When there are ongoing issues on a plant, some parameters display the current weather and an alarm sign.
   - **Yearly**: rank the KPIs of two different years.
2. **Date picker**: select two dates using the calendar picker. The first date is used to rank the KPIs of your portfolio against the second date.

ⓘ NOTE: You cannot pick dates when you are ranking KPIs in real time.

3. **Refresh**: click to refresh the data on display.
4. **Export**: click to export the data currently displayed on the list to a Microsoft Excel format. For more information, see Export Data to File.
5. Drag-drop area: click and drag a column header and drop it to this area to group the plants by one or more criteria.
6. **Column headers**: display the KPI name and the measure unit between brackets. Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon on any column header for advanced filtering. For further information on advanced filtering, see Advanced Filters
7. Ranking table: displays portfolio KPIs, with the total of each column at the bottom of the table. Select one or more cells, right-click and then click the icon to display the KPIs in the Data Viewer module or click the icon to display them in the Linear Chart Viewer module.

   There are four display modes:

   - 

Simple: displays data as text.
   - 

Status: displays data in a colored status bar that expresses the performance as a percentage.
   - 

Health: displays data as text. Colored icons indicate the health of the device by comparing one KPI over two different periods. For more information, see the Health icons section.
   - 

Weather: displays data as text. Weather icons display the current plant weather. The weather icons are only displayed in Real Time tables.

Health icons

The health icons display a range of production values as percentages of the reference value for an element or a plant.

Icon	Description
	00.01% above the reference value
	02.01% above the reference value
	13.61% above the reference value
	34.01% above the reference value
	00.01% below the reference value
	02.01% below the reference value
	13.61% below the reference value
	34.01% below the reference value

Reports module

To access the Reports module, click the  icon on the Upper Bar.

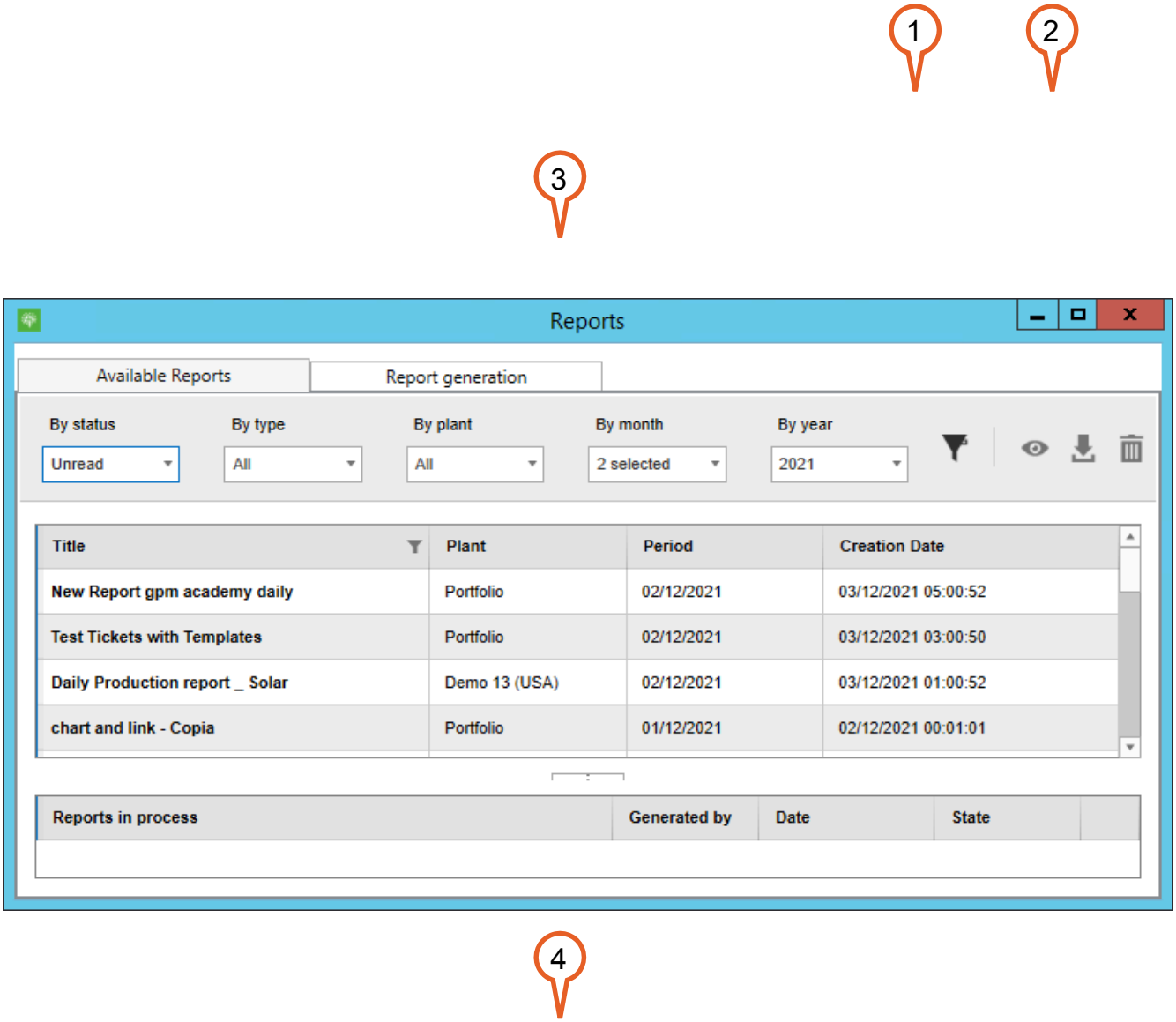
The interface of the Reports module consists of two tabs:

- **Available Reports**: access existing reports.
- **Report Generation**: create new reports.

Available Reports

This tab allows you to access existing reports. By default, the list displays unread reports from the current year and the most recent reports.

Available Reports tab



1. **Template filters**: click to select the filtering criteria from the drop-down lists and

click the icon to display reports that match the criteria on the list.

2. **Action buttons**: click to perform actions on the selected reports:
   - **Open** the selected report. Each report you select opens in a separate window.
   - **Download** the selected report. Each report you select opens a separate download dialog.
   - **Delete** the selected report.
3. Reports list: displays available reports. Select one or more reports to perform actions on them.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. click the icon on the **Title** column header for advanced filtering. For more information, see Advanced Filters.
4. Reports in process: displays reports that are being generated. You can cancel the generation fo a report by clicking the icon on the right side of the list.

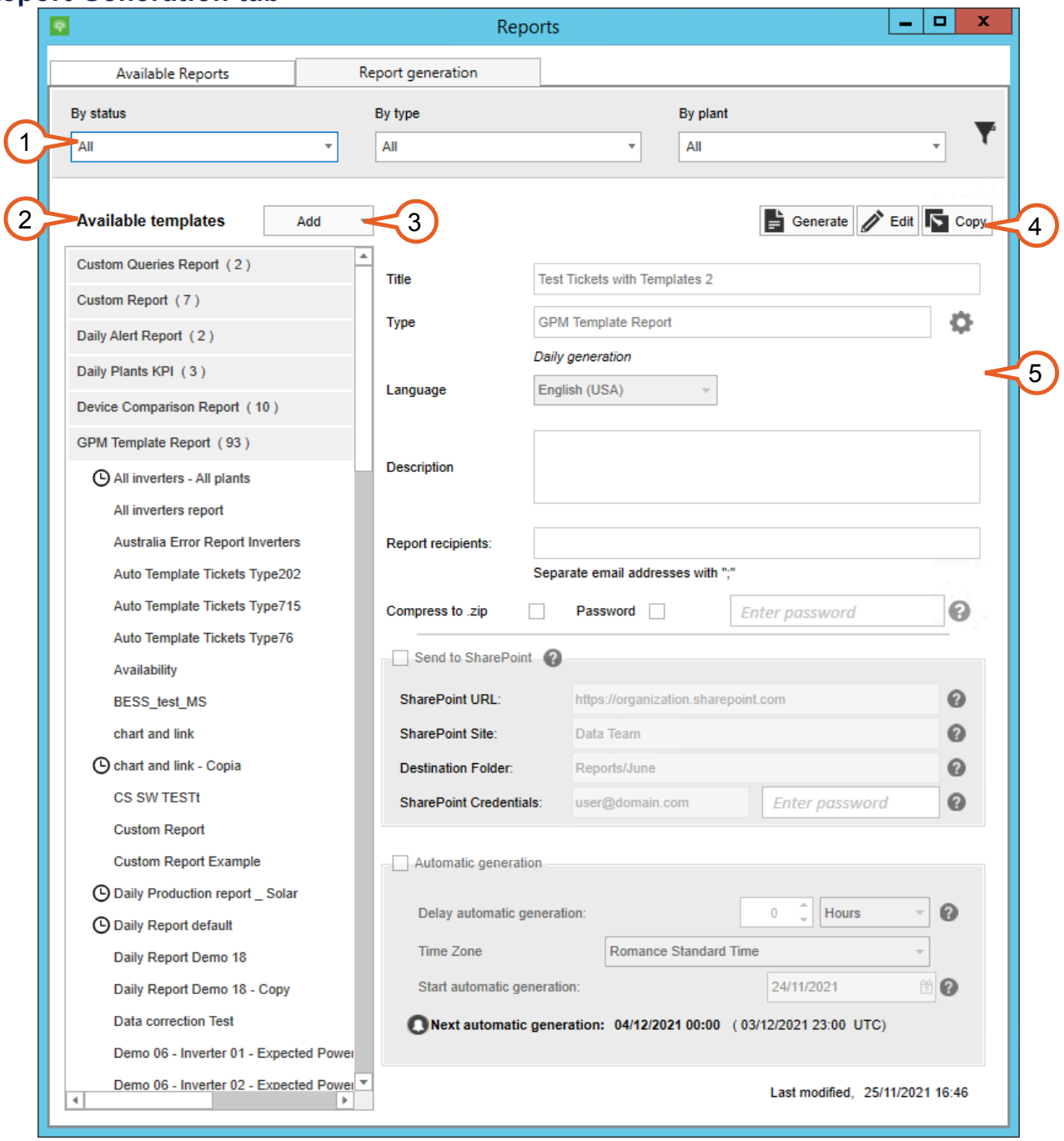
   Click on a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers.

Report Generation

On this tab, you can generate new reports, schedule automatic report generation, and edit existing reports. You can create new reports based on templates that are configured for you.

When you save your report, it becomes available in the **Available templates** panel in the section of the template on which it is based.

Report Generation tab



1. **Template filters**: click to select the filtering criteria from the drop-down lists and click the icon to display reports that match the criteria on the **Available**

templates panel.

2. Available templates: displays the available reports, organized by report type. Click on a template to edit it.
3. **Add template**: click to select a template from the drop-down list. For further information, see Create Reports using the GPM template.
4. **Action buttons**: click to perform actions on templates or reports:
   - **Generate** one of the selected reports manually. This button is only available for saved templates.
   - **Enable editing** for a selected report. This button is only available for saved templates.
   - **Create a copy** of the selected report. This button is only available for saved templates
   - **Save** your changes. This button is only available for unsaved reports.
   - **Cancel** your changes. This button is only available for saved reports.
   - **Delete** the selected report. This button is only available for saved reports.
5. Report settings: configure the settings for the report. This template becomes available when you add new report or when you are editing one of the available templates.

SCADA layout module

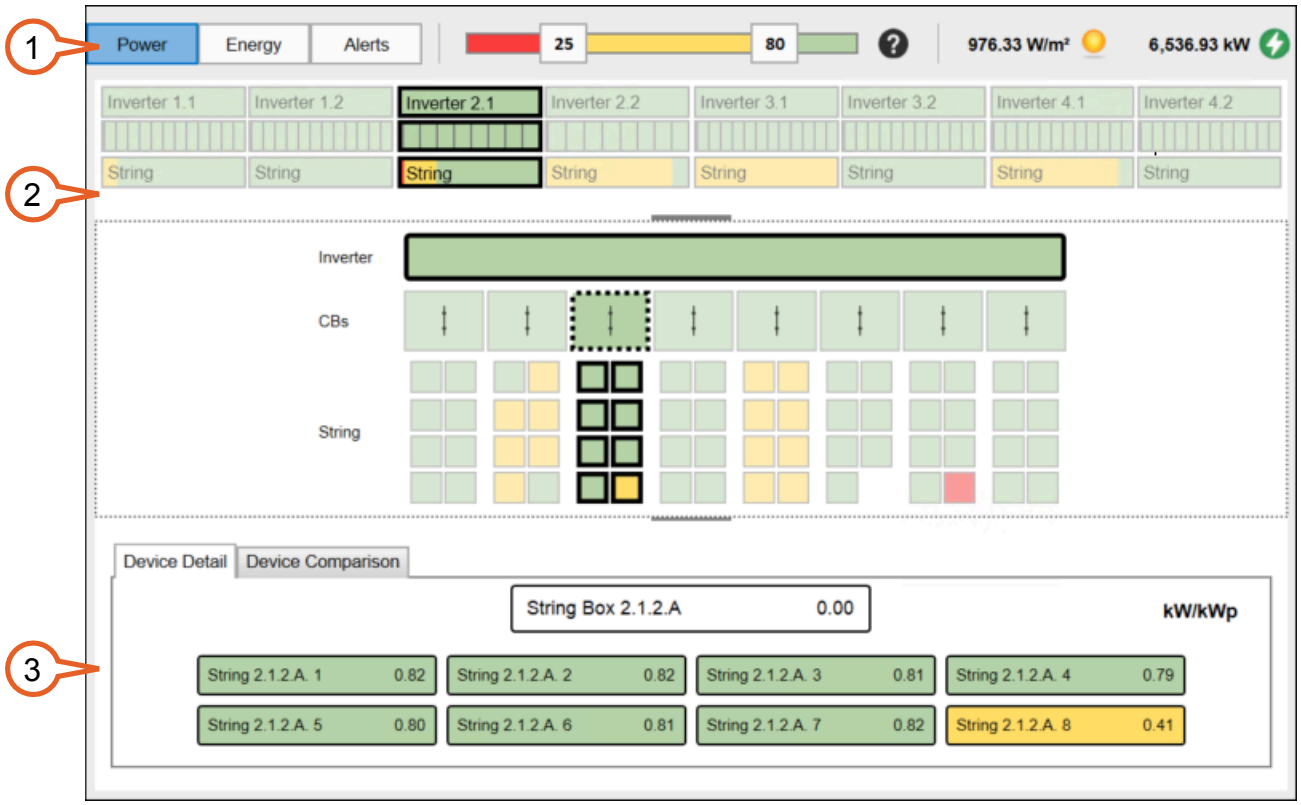
The Supervisory Control and Data Acquisition (SCADA) layout modules provides a global view of the plant efficiency and of the real-time status of the plant devices.

The SCADA Layout module uses color-coded cells to represent the performance of the devices. Each cell represents one element, and the cell size depends on the position of the element in the hierarchy. The performance of devices is calculated using the value of the best performing device as a reference and then expressed in percentage. The reference device can either be the best performing device of the same type or the best performing sibling device. The SCADA Layout also offers tools to analyze a single device and to compare multiple devices.

This module has three modes to analyze efficiency:

- **Power**: performance is calculated by comparing the device performance against a reference device for each device type. The reference is the latest and highest normalized power value for each device type.
- **Energy**: performance is calculated by comparing the device performance against a reference device for each device type. The reference is the highest daily normalized energy value for each device type.
- **Alarms**: each device is colored with the color of its most representative alarm that is active at the time. There is no comparison between devices in this mode.

SCADA layout



1. Main Menu
2. Element Hierarchy
3. Element Analysis

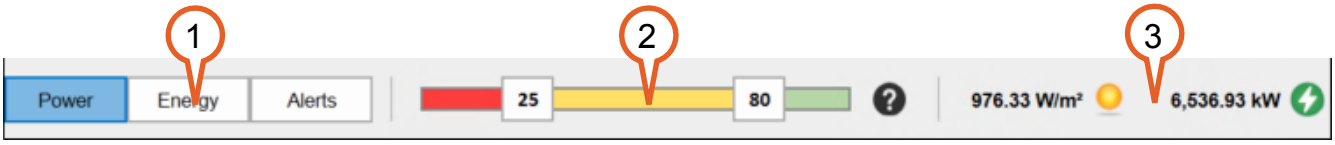
Color	Performance
Dark green	The device used as a reference to calculate the performance of other devices.
Green	The device is performing above the highest threshold.
Yellow	The device is performing between the lowest and highest thresholds.
Red	The device is performing below the lowest threshold.
White	The device is not communicating with the system and data cannot be retrieved.

Main Menu

The main menu is where you can change the SCADA Layout mode, customize the threshold for device comparison, see the active alarms, and see the plant weather and the latest total power or energy produced by the plant.

> ⓘ NOTE: The options available on the user interface vary depending on the mode that you select.

Main menu



1. Mode selector: click to switch between **Power**, **Energy** and **Alarms**.
2. Slider bar: click and drag the sliders to define the thresholds for performance. Click the ? icon to display the legend.

   > ⓘ NOTE: The slider bar is only available in Power and Energy modes.

   OR:

   Active alarms: displays a count of the active alarms, divided by type. Click the ? icon to display the legend.

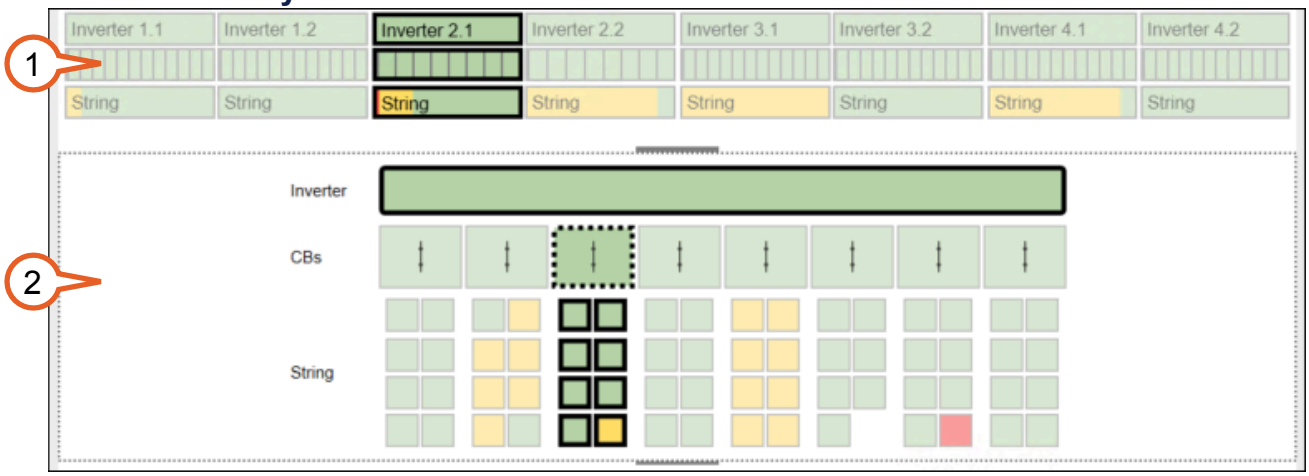
   > ⓘ NOTE: This is only available in Alarm mode.

3. Plant information: displays general information about the plant. The information depends on the selected mode:
   - **Power**: total plant irradiance, plant weather, and total plant power.
   - **Energy**: total plant insolation, plant weather, and total plant energy.
   - **Alarms**: available peak power percentage and the number of dataloggers currently communicating with the system.

Element Hierarchy

The Element Hierarchy is a visual representation of the plant production hierarchy with parent sets of elements on top, followed by child sets of elements. For example, inverters are on the top lines, followed by string boxes, which in turn are followed by string.

Element hierarchy



1. Plant units: displays the high-level hierarchy by units. For every unit, you can see the number of levels and use the color-coding to understand the general performance of each level. The last hierarchical level can be customized to condense all the information of its devices in a single cell. This cell is filled proportionally with the device performance color coding.

   Click a unit to display its devices in the Plant elements area.

   ⓘ BEST PRACTICE: We recommend activating the Plant Units area for plants with a high number of devices, when it is needed to divide the plant in smaller units.

2. Plant elements: displays the devices and their granular hierarchy. Click a device to display its details in the Device Details area or to add it to the Device Comparison areas. The selected device is highlighted with a dotted line and its parent and child devices with a full line.

   ⓘ BEST PRACTICE: Switches cannot be compared and have only two possible statuses:

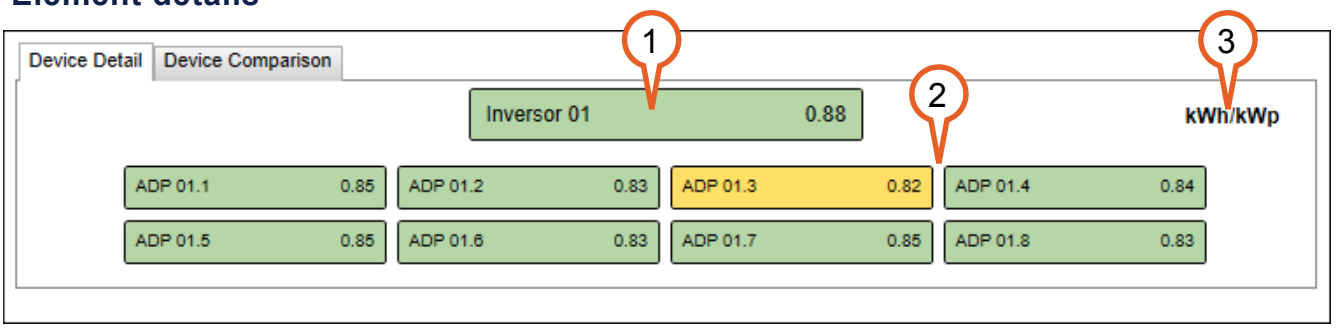
   - The switch is open.
   - The switch is closed.

Element Analysis

The Device Analysis area consists of two tabs where you can see the device details and compare devices between them:

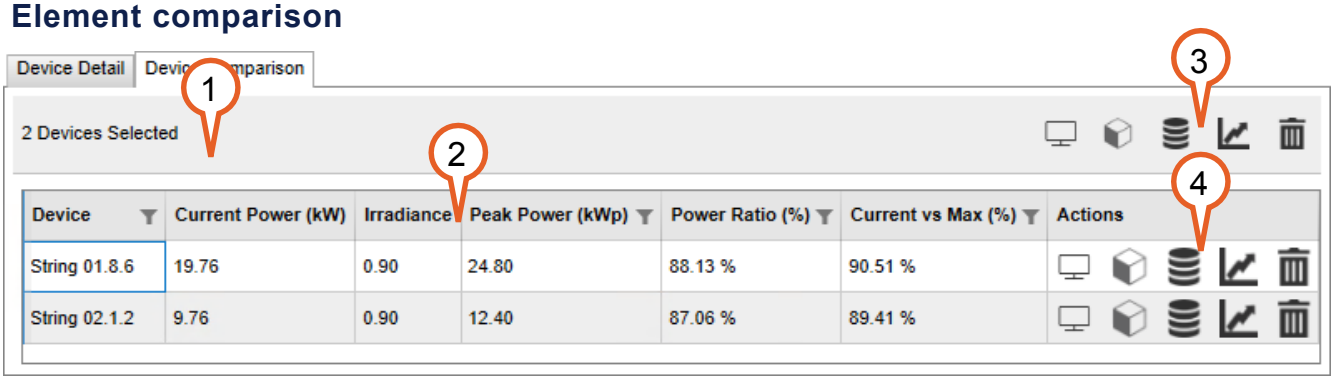
- Element details
- Element comparison

Element details



1. Selected element: displays the selected element as a cell. Its efficiency is expressed by the color-coding and by the value on the right side of the cell.
2. Child elements: the first level of child elements for the selected element are displayed as cells. Their efficiency is represented by the color-coding and by the value on the right side of the cell.
3. Performance unit: the unit used to express the element performance.

Element comparison



1. Element count: displays the number of selected elements.
2. Comparison table: displays selected elements. You can click any header to sort the table by that column.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon on any column

header for advanced filtering. For further information, see Advanced Filtering.

3. Group actions: click to open and compare the selected elements in other modules:
   - **Live Viewer**
   - **Element Viewer**
   - **Data Viewer**
   - **Linear Chart Viewer**

   Click the icon to delete all elements from the table.
4. Element actions: click to open the element in other modules:
   - **Live Viewer**
   - **Element Viewer**
   - **Data Viewer**
   - **Linear Chart Viewer**

   Click the icon to delete all elements from the table.

Scatter Plot Chart

The Scatter Plot Chart Viewer is a tool that allows you to create queries and analyze your portfolio's data using scatter plos charts. A scatter plot chart allows you to display the relationship of two sets of data by placing them along two axes. The results of your queries can be saved for further use or exported to other views.

You can add queries to the Live Chart Viewer from other areas of the user interface or create queries directly from the Scatter Plot Chart Viewer module.

To access the Scatter Plot Chart, click the icon on the Upper Bar, then click the icon.

Scatter Plot Chart

1. Query Period: click the icon to expand the menu and customize the data granularity and time range. For information, see the Query Period section below.
2. Add Parameters:click the icon to expand the menu and add parameters to the table. For more information, see the Add Parameters section. below.

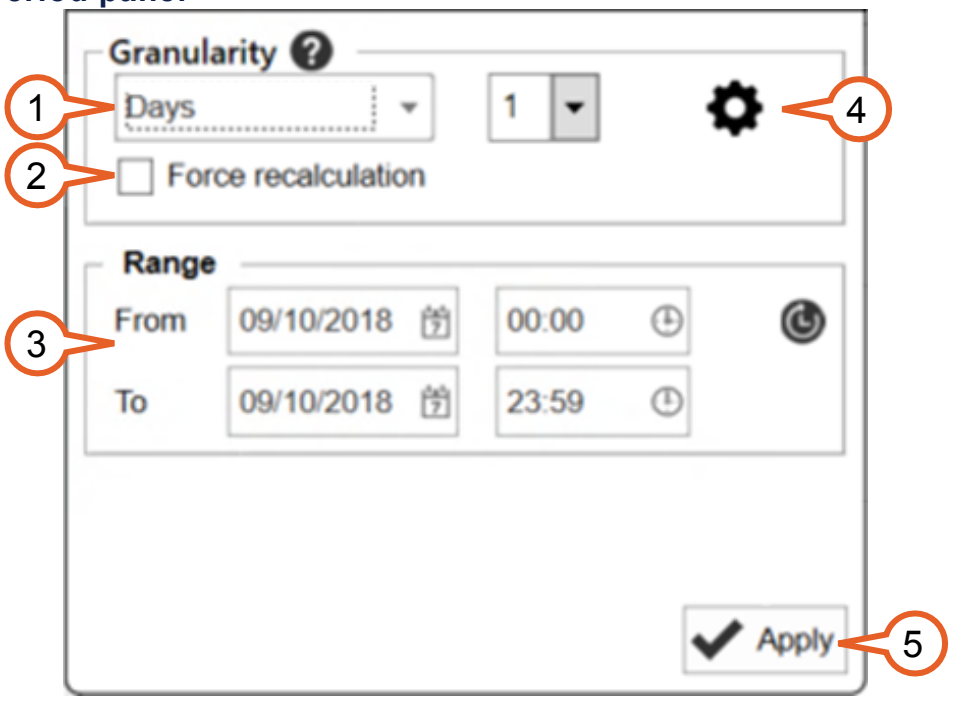
3. Action Buttons: click to perform quick actions:
   - **Open** the Data Filtering window to filter the data on display.
   - **Toggle zoom** on and off. When zoom is enabled, use the mouse scroll or drag and drop the area of the chart that you want to zoom.
   - **Save** the current query as a favorite.
   - **Load** a favorite query.
   - **Display** the selected parameters in the Linear Chart Viewer module.
   - **Display** the parameters in the Live Viewer module.
   - **Export** the query to the clipboard or to a file. For further information, see Export Data to File.
4. Current Parameters: click the icon to expand the menu and manage current parameters. You can change the display name and color for a parameter. You can also decompose parameters. For more information, see the Current Parameters section below.
5. Chart: displays data as a plot. Place your cursosrover a dot to display a tooltip with the parameter name and the value of each axis.

   Click the labels of the **X-axis** or the **Y-axis** to customize their ranges and auto-scale them.
6. Legend: displays the results of your query as a table. You can hide data from the chart and customize the appearance of single parameters. For further information, see the Legend section below.
7. **Period browser**: browse the time range using the arrows. The time range displayed is based on the query period that you selected.

Query Period

The Query Period panel lets you customize the data granularity and time range of your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Query Period panel



1. **Data granularity**: click to access advanced settings for data granularity.
2. **Force recalculation**: click to manually force a recalculation of the data if the values have changed you do not want to wait for the next automatic recalculation to see them. The system automatically calculates values at recurring intervals and stores them in the database for faster access.

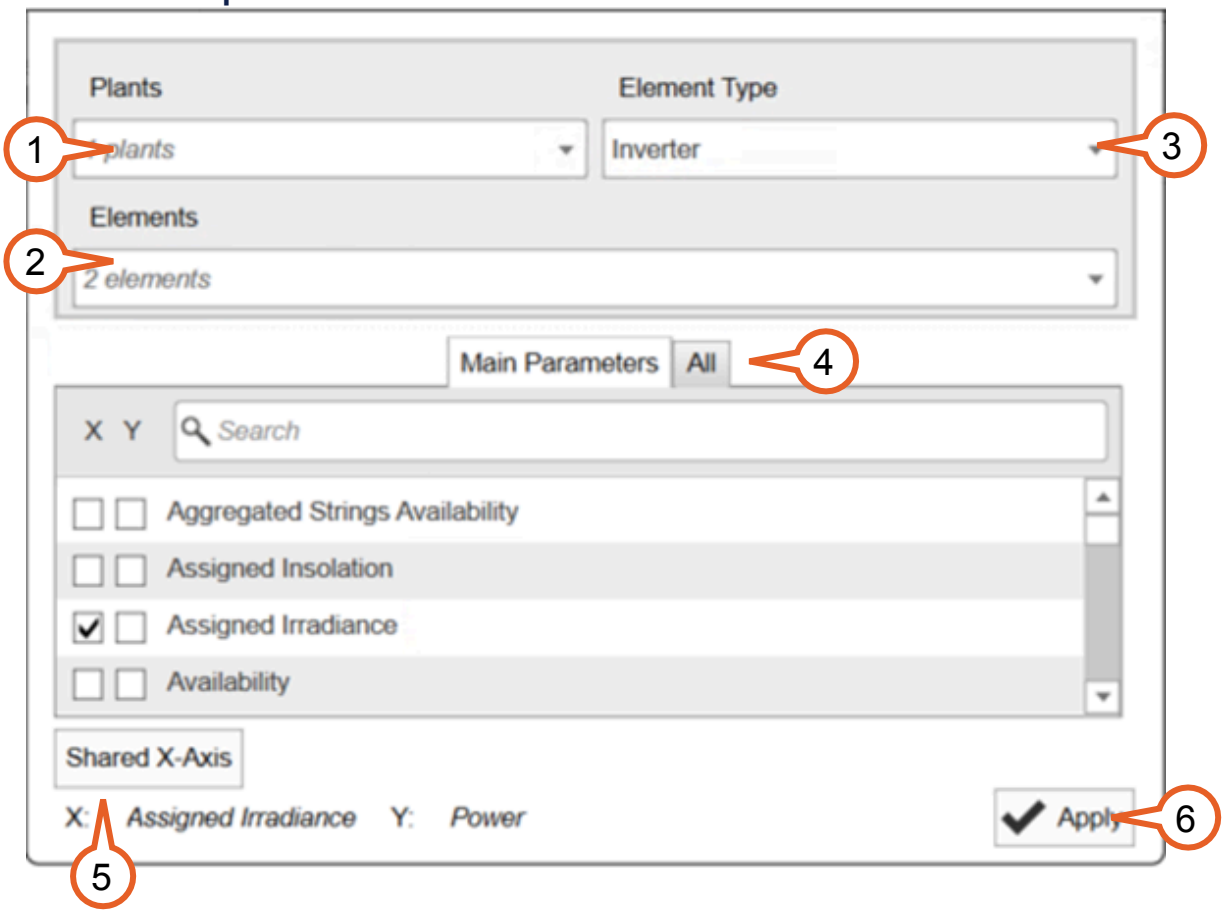
   This option is only available when the selected granularity is **Days**, **Months**, or **Years**.

   ⓘ NOTE: Recalculations may take several minutes.
3. **Date and time range**: select the date and time range using the calendar picker. Click the ⏲ icon to set the range to the default period.
4. **Data granularity settings**: click to access advanced settings for data granularity.
5. **Apply**: click to apply your settings to the query.

Add Parameters

The Add Parameters panel lest you add parameters to your query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons.

Add Parameters panel



1. **Plants**: select one or more plants from the drop-down list. You can use the *Search* field to refine the drop-down list results.
2. **Elements**: select one or more elements from the drop-down list.

   ⓘ NOTE: You can only select elements after selecting a single plant. If you select multiple plants, this option is disabled.

3. **Element type**: select one or more element types from the drop-down list.

   ⓘ NOTE: You can only select element types after selecting multiple plants.

4. **Parameters**: click to select one or more parameters or series from one of the tabs. You can use the *Search* field to refine the drop-down list results.

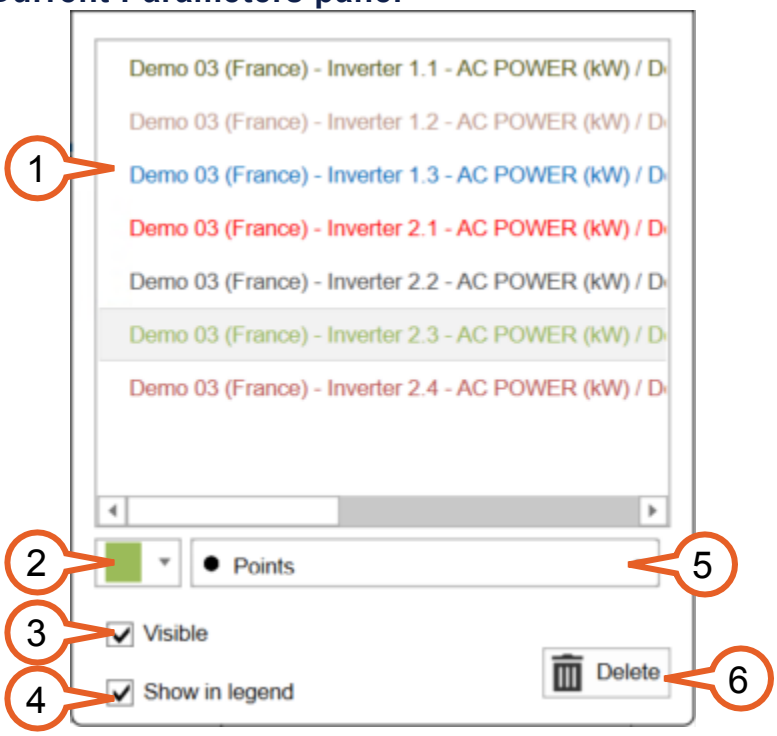
- **Main Parameters**: lists the most relevant parameters that are configured for the selected element.
- **All**: lists all the parameters that are retrieved from the selected element.

5. **Shared X-axis**: click to select the X-axis parameter if you want to compare multiple parameters on the Y-axis against a single parameter on the X-axis.
6. **Apply**: click to apply your settings to the query.

Current Parameters

The Current Parameters panel lets you the manage parameters included in a query. You can expand and collapse the panel by clicking the ⌄ and ⌃ icons. You can also change the display name and the color of a parameter.

Current Parameters panel

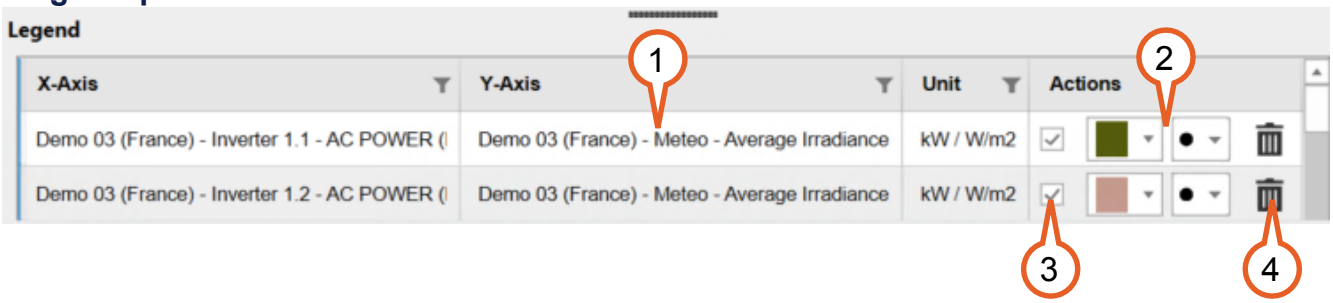


1. Parameters: lists the parameters currently displayed on the table. Select one or more parameters to customize their appearance. To select multiple parameters, use CTRL+Click.
2. **Point color**: select a color for the parameter from the drop-down list.
3. **Visible**: toggle the checkbox to display or hide the selected parameter from the chart.
4. **Show in legend**: togglethe checkbox to display or hide the selected parameter from the legend.
5. Point shape: select the shape for the data points from the drop-down list. Available options are **Points**, **Rectangles**, and **Diamonds**.
6. **Delete**: click to delete the selected parameters from the chart.

Legend

The legend panel provides quick access to the chart legend and displays the results of your queries as a table. You can filter and sort the data on display and customize the colors for the parameters.

Legend panel



1. Table: the name of the parameters used to plot the chart, by axis and by unit. Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon on any column header for advanced filtering. For further information on advanced filtering, see Advanced Filters.
2. **Point color and shape**: select the display color for the data points from the drop-down list.
3. **Display/Hide**: toggle the checkbox to display or hide the parameter from the chart.
4. **Delete**: click to delete the selected parameters from the chart.

Tickets module

The Tickets module contains the tools to manage and follow up on corrective and preventive maintenance duties for plants. Tickets can be single or recurring. Recurring tickets are called "series".

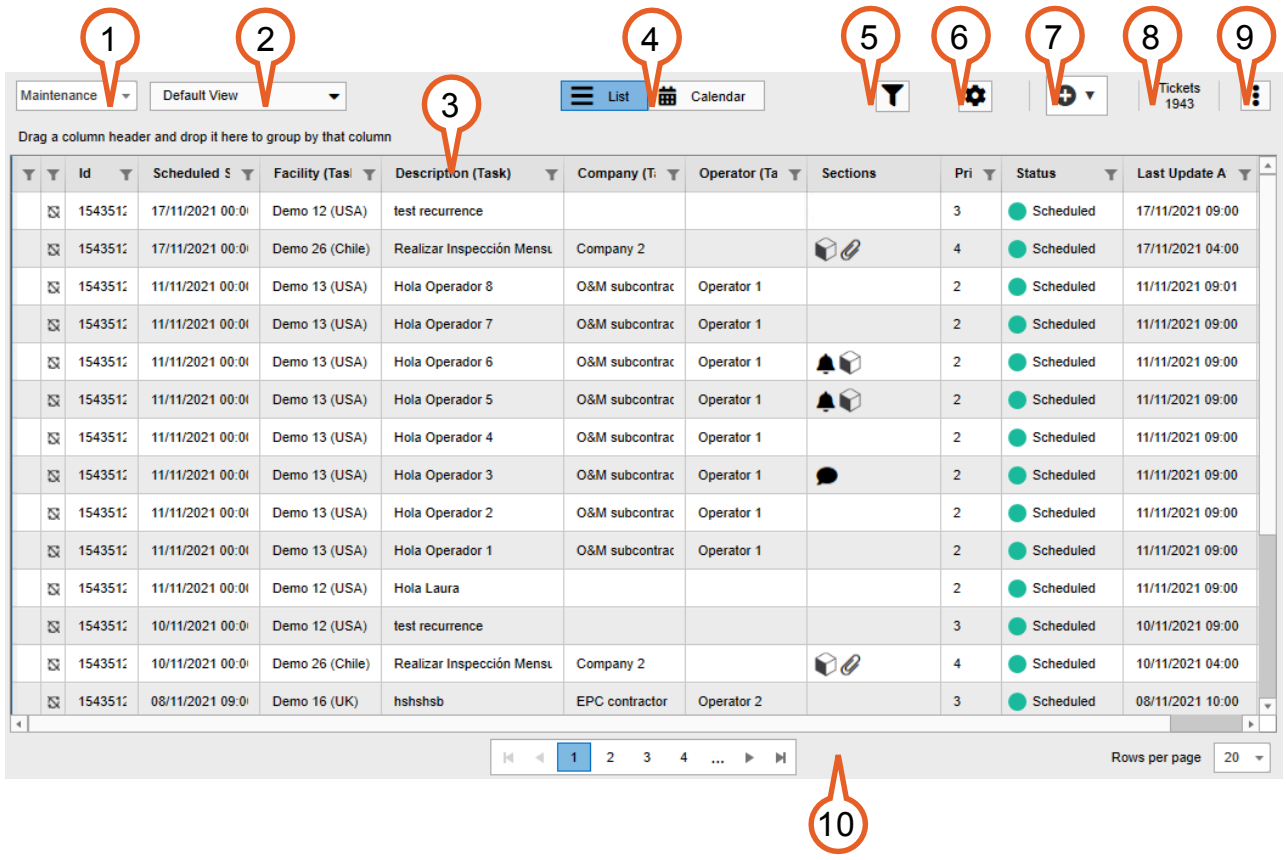
The tickets list displays detailed information for all the tickets currently in the system. You can display tickets in two modes: **List** or **Calendar**.

It is possible to configure custom views of the list, and you can also filter the tickets displayed in the view using different criteria and save the filter to reuse it as a default view. For more information, see the tasks to configure the tickets module.

You can access the Context Menu for tickets by selecting one or more tickets and right-clicking on it. For more information, see Tickets Context Menu below.

> ⓘ NOTE: When you select multiple tickets, certain menu options are disabled or limited.

Tickets module: List view

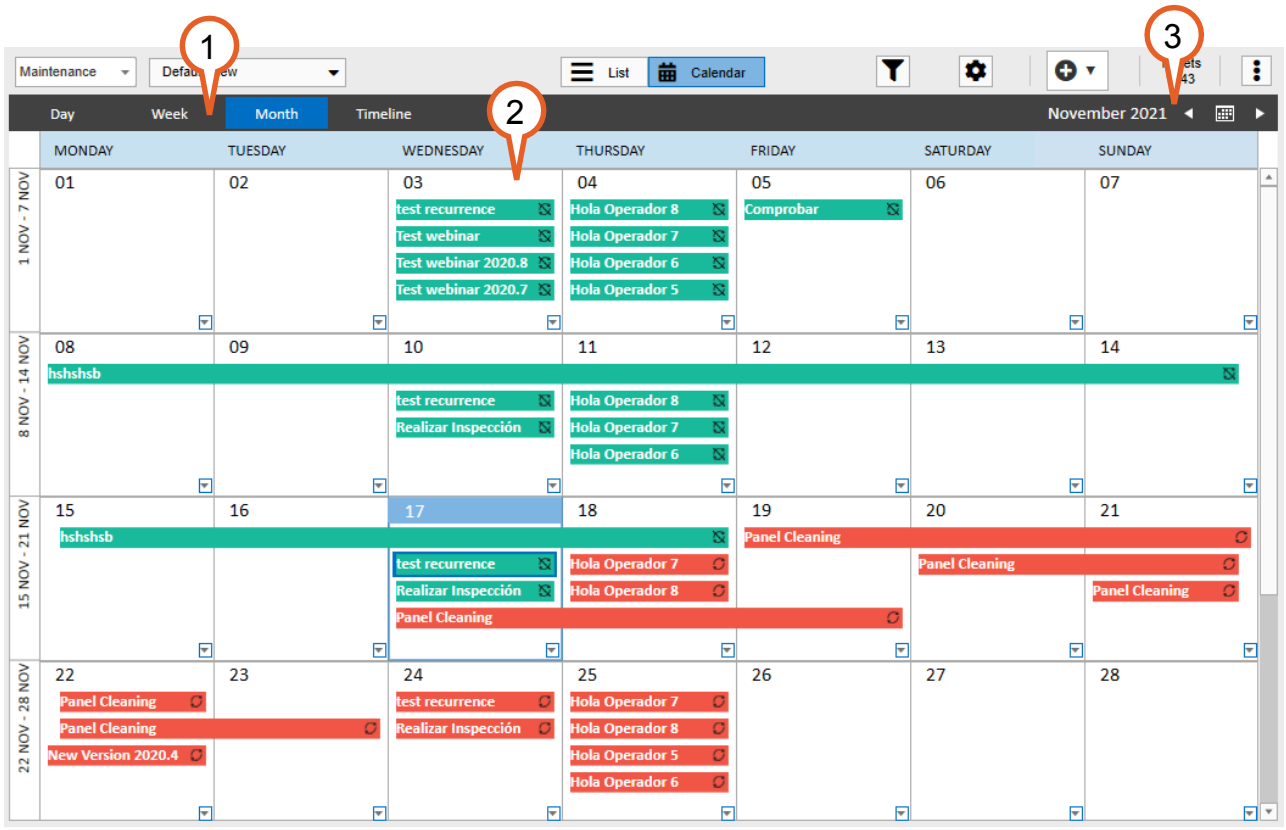


1. **Ticket selector**: select the type of ticket to display in the Ticket List (for example, "Maintenance").

2. **Available views**: select a view from the drop-down list to display the tickets that match it. You can also save the current filtering criteria as a new view. For more information about creating new views or editing existing views, see the instructions to configure the tickets module.
3. Tickets list: double-click a ticket to open the Edit Ticket dialog. Right-click a ticket to open the context menu and perform quick changes to the ticket. For further information, see Context Menu.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. Click the icon on any column header for advanced filtering. For further information, see Advanced Filters.
4. View mode: switch between view modes.
5. Advanced filter: click the icon to expand or collapse the filtering options and refine filtering criteria. For further information, see Tickets Module Filtering Options.
6. Display settings: click to customize the columns displayed on the work order grid. You can add or remove columns, rearrange them, or customize the column header text. For more information, see the instructions to configure the tickets module.
7. New ticket: click and select a ticket type to create a new ticket.
8. Ticket count: displays the total count of the tickets on the table.
9. Options: click to display the available options:
   - Export data: export the tickets currently displayed on the list to a file in Microsoft Excel format. For further information, see Export data to file.
   - History: display the history of the selected ticket.
   - Export template: download the ticket template to create tickets in bulk using your spreadsheet editor.

     ⓘ NOTE: This action requires the administrator password.
10. Pagination: define the number of tickets to display on each page and browse the pages using the arrows.

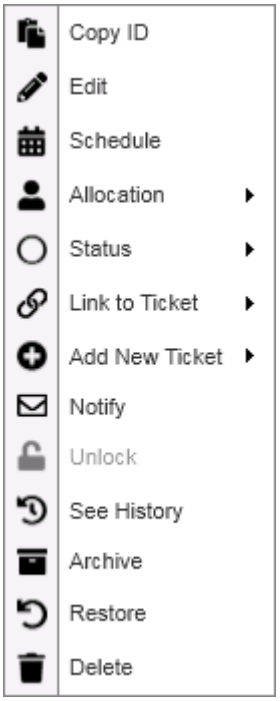
Tickets module: Calendar view



1. Range selector: select the calendar date range or display tickets as a **Timeline**.
2. Calendar: double-click a ticket to edit it.
3. Date selector: browse to the previous or next period.

Context menu

Context menu



Option	Description
Copy ID	Click to copy the ticket ID. This option is only available when you select a ticket that is not in a series.
Edit	Click to edit the ticket. You can only edit one ticket at a time from the UI. It is possible to edit tickets from the same template in bulk. To do this, you must export them as an XLS file, edit the values and import the modified file. For further information, see Export Data to File and Import File.
Schedule	Click to change the scheduled start and scheduled end of the task inside a ticket. This option is only available when you select one or more non-recurring tickets.
Allocation	Assign the ticket to another operator.

	This option is only available when you select one or more non-recurring tickets.
Status	Change the status of the selected tickets. This option is only available when you select one or more non-recurring tickets.
Link to Ticket	Select a relational link between a single ticket and another ticket. For further information, see Link a Ticket to Another Ticket.
Add New Ticket	Create a new ticket with a relational link from a single ticket. For further information, see Create a New Ticket from an Existing Ticket.
Notify	Click to open the **Notification Settings** dialog and send an email notification regarding the selected ticket. This option is only available when you select one or more non-recurring tickets. When multiple non-recurring tickets are selected, you can only send a notification without customizing the content.
Unlock	Click to force the unlocking of a ticket that was locked from the Offline App. ⓘ NOTE: The unlock functionality is available depending on your GPM SCADAconfiguration.
See History	Click to see the history of changes to the selected non-recurring ticket or tickets.
Archive	Click to archive the selected ticket. Archiving is useful when a ticket is currently not required but may need to be restored or viewed later. For further information on how to archive tickets, see Archive a Ticket. Archived tickets are hidden from the ticket list, but you can display them by enabling the **Archived** toggle in the filtering options panel. For further information on the available filtering options, see Tickets Module Filtering Options.

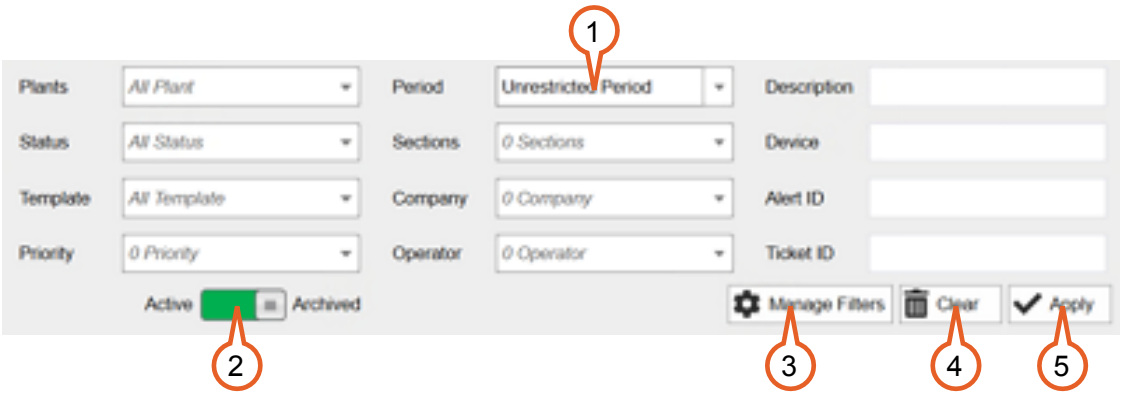
Restore	Click to restore the selected tickets. This button is only available for archived tickets. For further information on how to restore tickets, see Restore Archived Ticket.
Delete	Click to delete the selected tickets. Deleting is only possible after a ticket is archived and permanently removes the ticket from the application. For further information on how to restore tickets, see Delete Ticket.

Filtering options

The Tickets module has additional filtering options located in a panel that you can access by clicking the icon in the upper-right area of the user interface. This section lists the filtering options available by default in the Tickets module.

After selecting the filtering criteria, click **Apply** to display the relevant tickets in the ticket list.

Filtering options



1. **Fields**: click to select the filtering criteria from the drop-down menus.
2. **Active/Archived**: toggle between active or archived tickets.
3. **Manage filters**: click to add or remove custom filtering criteria from the filtering options.
4. **Clear**: click to clear the filtering criteria and display all available tickets on the list.
5. **Apply**: click to apply the filtering criteria and display the relevant tickets on the list.

Field	Description
Plants	Filter by the plant to which the tickets belong.
Status	Filter tickets by their status.
Template	Filter tickets by their template.
Priority	Filter tickets by their priority.
Period	Filter tickets by period. The period is calculated from the Scheduled Start time stamp. The case-sensitive toggle is not available when you are filtering numerical values.

Sections	Filter tickets by the presence of specific ticket sections.
Company	Filter tickets by the company to which they belong.
Operator	Filter tickets by the operator assigned to them.
Description	Filter tickets by the content of their descriptions.
Device	Filter tickets by the name of their linked devices. You can enter multiple device names, separated by a comma or enter some text between two "%" symbols to use partial matching. For example, "%ren%" is a partial match for the term "renewable".
Alarm ID	Filter tickets by the ID of the alarms linked to them.
Ticket ID	Filter tickets by their Ticket ID. You can enter multiple Ticket IDs, separated by a comma.

Configure the Tickets module

You can perform several tasks to configure the views of the tickets table in the Tickets module:

- Create a new view of the tickets table.
- Edit existing views:

  > ⓘ NOTE: You can only edit views that you have created yourself.

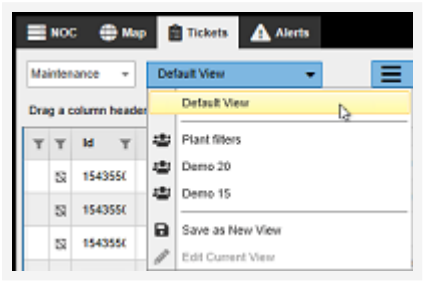
  - Edit settings.
  - Edit filters.
  - Edit columns.

Create new views for the Tickets list

To create a new view for the Tickets list, follow these steps:

1. In the Tickets module, click the drop-down menu of available views and select the **Default View**.

Available views menu



2. Click the drop-down menu of available views and select **Save as new view**.

   Result: The **New view** dialog appears:

> ⓘ NOTE: The new view inherits the filters and settings of the view you selected in Step 1. If you select a view other than the Default view, the filters and settings will match those of the selected view.

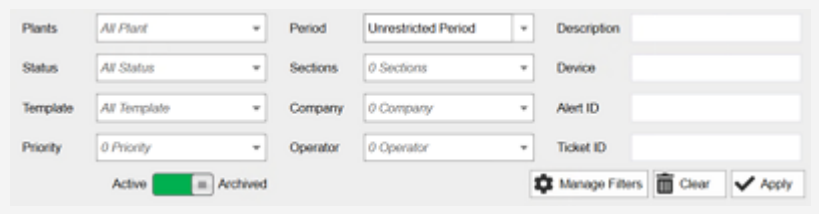
New view dialog

3. Enter the *View name*.
4. (Optional) Toggle on **Private** to make the view available only to your user account.
5. (Optional) Check the **Default view checkbox** to set this as the default view when you open the Tickets module.
6. Click **Save**.

7 Click the ▼ icon.

Result: The filtering options appear:

Filtering options

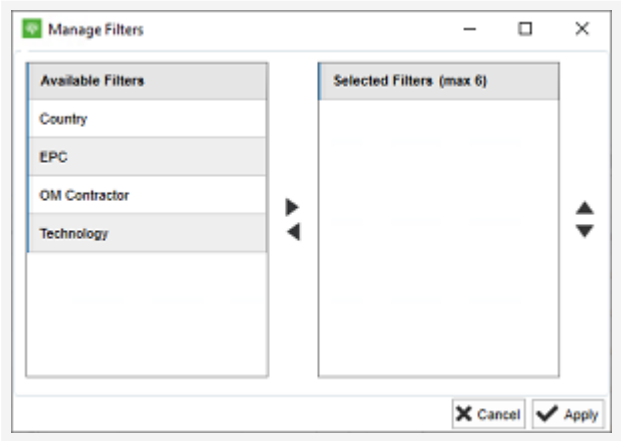


ⓘ NOTE: For detailed information, see the section on filtering options.

8 (Optional) Click **Manage filters** to open a dialog to see more available filters:

Manage filters dialog

ⓘ NOTE: The available filters depend on your product configuration.



a Select a filter from the **Available filters** panel and click the ▶ icon to move it to the **Selected filters** panel.

b (Optional). To change the order of the selected filters, in the **Selected filters** panel, select a filter and click the ▲ icon to move it up, or the ▼ icon to move it down.

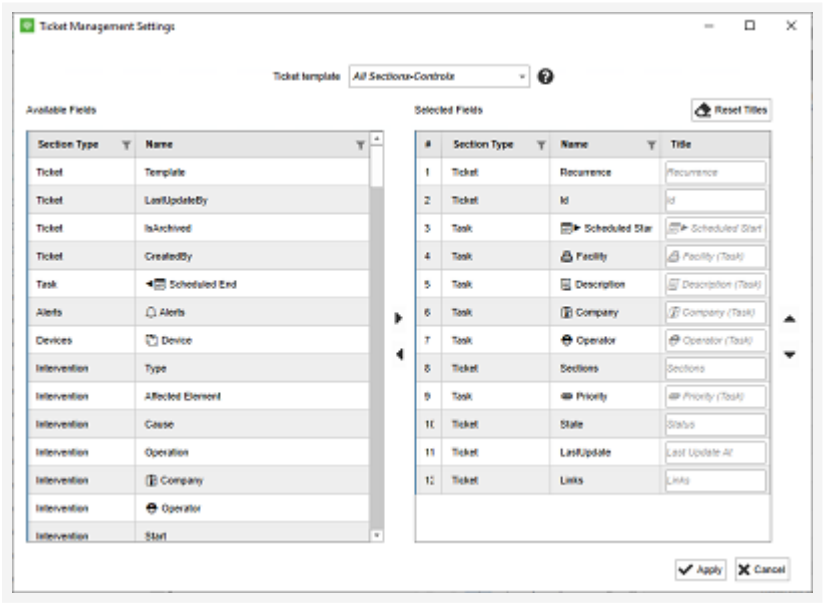
c Click ✓ **Apply**.

Result: The selected filters appear in the **Filtering options**.

9 Click the ⚙ icon.

Result: The **Ticket management settings** dialog opens.

Ticket management settings dialog



10. Configure the columns for the Tickets table:

    a. In the **Available fields** panel, select the columns you want to display on the Tickets table and click the ▶ icon to move them to the **Selected fields** panel.

    > TIP: Hold CTRL to select multiple options at once.

    b. (Optional). To change the order of the selected filters, in the **Selected filters** panel, select a filter and click the ▲ icon to move it up, or the ▼ icon to move it down.

    c. Click ✓ **Apply**.

    Result: The changes are applied and the table displays the selected columns in the order you defined.

11. Click the drop-down menu of available views and select **Edit current view**.

Result: The Edit view dialog appears:

Edit view dialog

12 Click **Save**.

Result

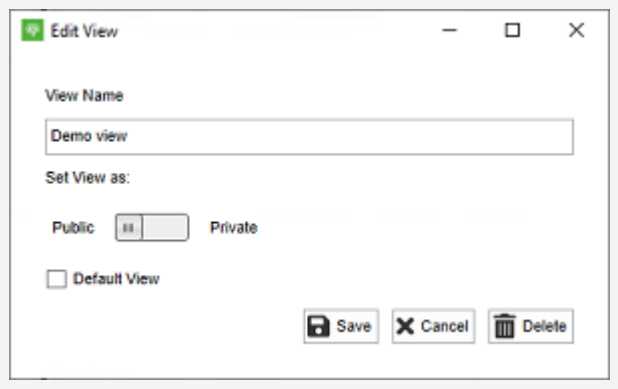
The new view is saved and added to the list of available views.

Edit settings for existing views of the Tickets list

To edit the settings for an existing view of the Tickets list, follow these steps:

1. In the Tickets module, click the drop-down menu of available views and select the view you want to edit.
2. Click the drop-down menu of available views and select **Edit current view**.
   Result: The Edit view dialog appears:

   ### Edit view dialog

3. (Optional) In the *View name field*, enter a new name for the view.
4. (Optional) To make the view available to all users, toggle on **Public**.
   Or: To make the view available only to your own user account, toggle on **Private**.
5. (Optional) Check the **Default view checkbox** to make this the default view when you open the Tickets module.
   Or: Uncheck the **Default view checkbox** if you do not want this to be the default view when you open the Tickets module.
6. Click **Save**.

Result

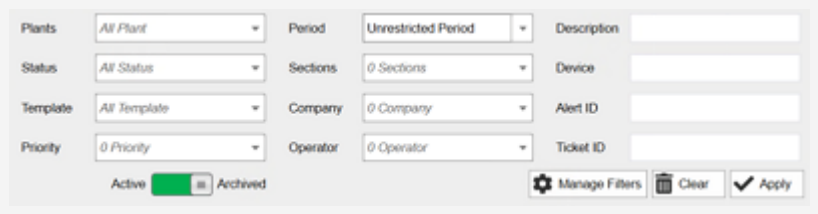
The settings are saved and applied to the view.

Edit filters for existing views in the Tickets list

To edit the filters for an existing view in the Tickets list, follow these steps:

1. In the Tickets module, click the drop-down menu of available views and select the view you want to edit.
2. Click the ▼ icon.
   Result: The filtering options appear:

Filtering options

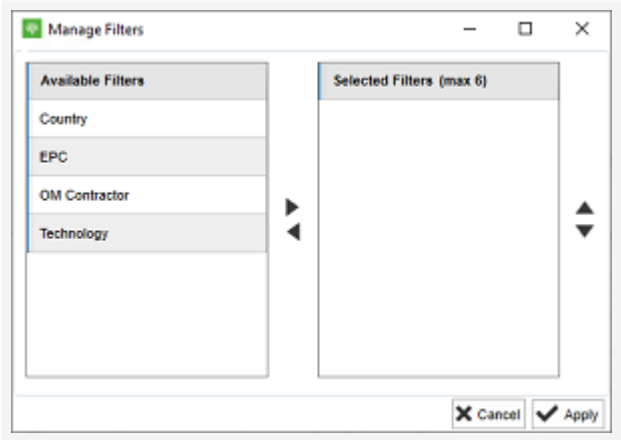


> ⓘ NOTE: For detailed information, see the section on filtering options.

3. (Optional) Click **Manage filters** to open a dialog to see more available filters:

Manage filters dialog

> ⓘ NOTE: The available filters depend on your product configuration.



   a. Select a filter from the **Available filters** panel and click the ▶ icon to move it to the **Selected filters** panel.
   b. (Optional). To change the order of the selected filters, in the **Selected filters**

panel, select a filter and click the ▲ icon to move it up, or the ▼ icon to move it down.

c. Click ✓ **Apply**.
Result: The selected filters appear in the **Filtering options**.

4. Select the filters you want to apply to the list:
   a. Open the **Plants** drop-down menu and select one or more plants from which you want to see tickets.
   b. Open the **Status** drop-down menu and select one or more statuses (for example, **Open**).
   c. Open the **Template** drop-down menu and select one or more templates.
   d. Open the **Priority** drop-down menu and select one or more levels of priority (for exaple, **High**).
   e. Open the **Period** drop-down menu and select a time period (for example, **Current week**).
   f. Open the **Sections** drop-down menu and select one or more sections of the tickets that must be filled in (for example, **Note**).
   g. Open the **Company** drop-down menu and select one or more companies to which the tickets have been assigned.
   h. Open the **Operator** drop-down menu and select one or more operators to which the tickets have been assigned.
   i. Enter a *Description* to display only tickets that match that description.
   j. Enter a *Device* name to display only tickets that contain the specified device (for example, "Inverter" or "Generator").
   k. Enter an *Alarm ID* to display only tickets linked to the specified alarm.
   l. Enter a *Ticket ID* to display only tickets linked to the specified ticket.
   m. Toggle on **Active** to display only active tickets.
   Or: Toggle on **Archived** to include tickets that have been archived.
   n. In the **Order by** section, select whether to order the list by **Last update** (chronological) or by **Ticket ID** (numerical).
5. Click ✓ **Apply**.
Result: The filters are applied and the Tickets table displays the updated list.

> ⓘ NOTE: You can click 🗑 **Clear** at any moment to remove all the selected filters and revert back to the Default template.

6 Click the drop-down menu of available views and select **Edit current view**.

Result: The Edit view dialog appears:

Edit view dialog

7 Click **Save**.

Result

The changes are saved.

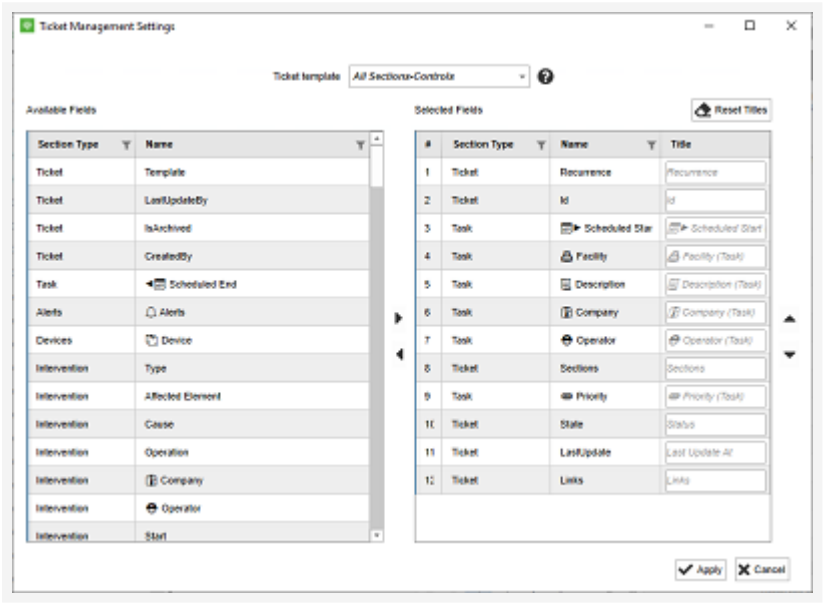
Edit columns for existing views in the Tickets list

To edit the columns for an existing view in the Tickets list, follow these steps:

1. In the Tickets module, click the drop-down menu of available views and select the view you want to edit.
2. Click the ⚙ icon.

   Result: The **Ticket management settings** dialog opens.

   ### Ticket management settings dialog

3. Configure the columns for the Tickets table:
   a. In the **Available fields** panel, select the columns you want to display on the Tickets table and click the ▶ icon to move them to the **Selected fields** panel.

      > ⓘ TIP: Hold CTRL to select multiple options at once.

   b. (Optional). To change the order of the selected filters, in the **Selected filters** panel, select a filter and click the ▲ icon to move it up, or the ▼ icon to move it down.
   c. Click ✓ **Apply**.

      Result: The changes are applied and the table displays the selected columns in the order you defined.
4. Click the drop-down menu of available views and select **Edit current view**.

Result: The Edit view dialog appears:

Edit view dialog

5 Click **Save**.

Result

The changes are saved.

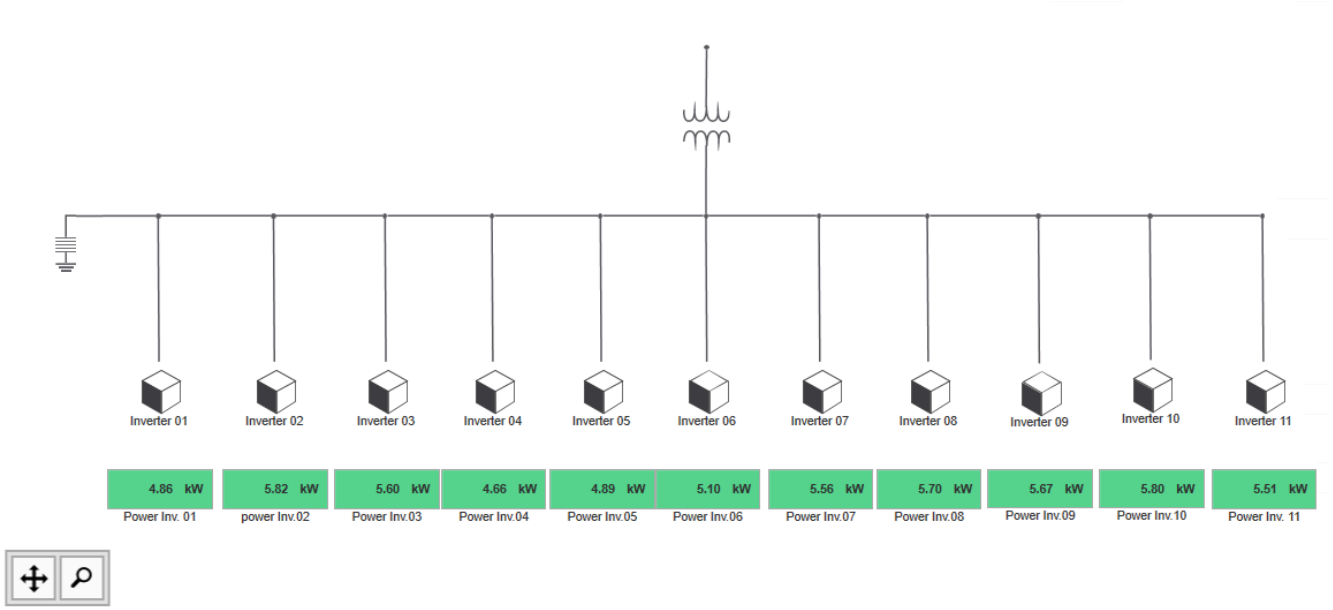
Vectorial Layout module

The Vectorial Layout module provides you with a multi-layered, custom vectorial schema to help you understand a plant's structure, its relationships, and the status of its elements. Within the Vectorial Layout, you can also send pre-defined commands to the hardware devices in your plant.

Access the module by clicking the **Vectorial** tab in the upper area of the screen.

> ⓘ NOTE: To customize your Vectorial Layout, contact your GPM representative.

Vectorial layout



Navigate the Vectorial Layout

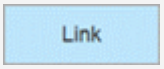
Action	Description
Mouse scroll	Use the scroll wheel on your mouse to zoom in and out of the Vectorial Layout. To zoom in, scroll up. To zoom out, scroll down.
Drag and drop	Click and hold on a point of the layout, then drag it anywhere to move around the interface.
Single click	Click on elements of the Vectorial Layout to interact with them. For further information, see the reference below.
Auto-fit	Click the icon at the bottom-left of the Vectorial Layout to automatically resize it and fit the screen.
Zoom	Click the icon at the bottom-left of the Vectorial Layout to set the zoom manually and preview the area being zoomed. Click the ✕ icon to close the zoom dialog.

Vectorial Layout elements

You can interact with some elements of the Vectorial Layout. This reference lists the elements with which you can interact.

> ⓘ NOTE: The background, element and icon colors in your vectorial layout are customizable. Contact your GPM representative if you want to customize them.

Element	Name	Description
	Alarm Counter	Counts the number of specific alarms for a specific set of devices. It is possible to set limits for the counter, so that its color changes from white to red when alarms exceed the configured number. When the alarms exceed the configured number and the counter turns red, you can click the box to display related alarms in the Alarms module.
	Command	Click to execute a command or a metacommand on one or more devices. If the command is configured to be executed on multiple devices, you can select the devices to which the command must be sent in the pop-up window. For more information, see Send a Command Sequence from the Vectorial Layout.
	Connection point	Displays the layout nodes.
	Data	Displays information that is retrieved from a parameter and is automatically refreshed at regular intervals. The box icon can be customized and up to three statuses can be set on every box. For example, the color can be set to change when a certain condition is met. Click a data box to display the values in the Live Viewer module.

	Element	Represents a physical device that is communicating with the application. If there is an active alarm on the element, an alarm icon is displayed next to it. Click an element to display further information in its relevant section.
	Icon	A static visual reference to a physical device in your plant. For further information, see Send a Command Sequence from the Vectorial Layout.
	Label	A text label used to identify elements.
	Line	Connects various physical or virtual elements in your setup. Lines can be assigned custom color-coding that changes when a specific condition is met.
	Link	Links the current layer to other layers in the layout. Click it to display the linked layer.
	Switch	Switches are elements that behave as a data box. This means that they can have values that change over time.

Vectorial icons

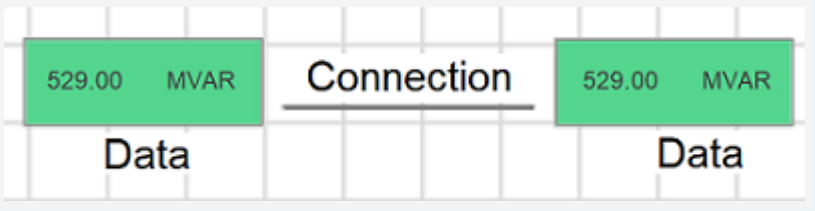
The application uses a default set of icons to identify specific device or element types in the Vectorial Layout.

Icon	Name
	Breaker
	Circuit breaker
	Current transformer
	Earth ground
	Fuse
	Generator
	Medium
	Meter
	Potential transformer
	Surge arrestor
	Transformer
	Transformer Type X
	Three-winding transformer

Configuring data objects and connection objects

You can configure data objects and connection objects using an XLSX template or the corresponding configuration window in the grid.

ⓘ NOTE: A **data object** is an element used to monitor a datasource value of a device, while a **connection object** is a line used to connect several objects.



Configuring data objects and connection objects using the XLSX template

I

- SourceType: Set the type of source the layer must use. You can enter one of the following options:
  - Datasource: It defines the type of source required in the *DatasourceId* column. When configuring a Monitored DS or Custom DS, you must provide a specific value for the *DatasourceId*. Additionally, you need to fill in the *ElementID* field in both cases.
  - ElementParameter: It indicates that the configuration of the object is determined by the *ElementParameterID* field. You must set a specific value for the element parameter during setup (parametrization).
- DataSourceComponentId

> ⓘ NOTE: This column can only be used in GPM Plus. Do not configure it as it will not display any data.

This is an example of how to configure layer settings for data and connection objects using the XSLX template:

Item	SourceType	ElementId	ElementParameterId	DatasourceId
DataElement PARAMETER	ElementParameter	247	76	0
DataElement MONITORED DATASOURCE	Datasource	247	76	0
DataElement CUSTOM DATASOURCE	Datasource	0		20941

Configuring data objects and connection objects in the grid

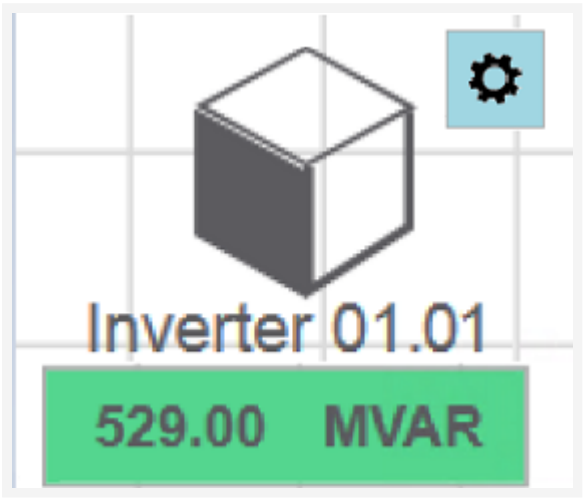
You can configure data objects and connection objects using the corresponding configuration window. You have to choose the desired element in the plant and assign to it parameters.

Configuring data objects in the grid

You can configure data objects using the corresponding configuration window.To configure data objects, follow these steps:

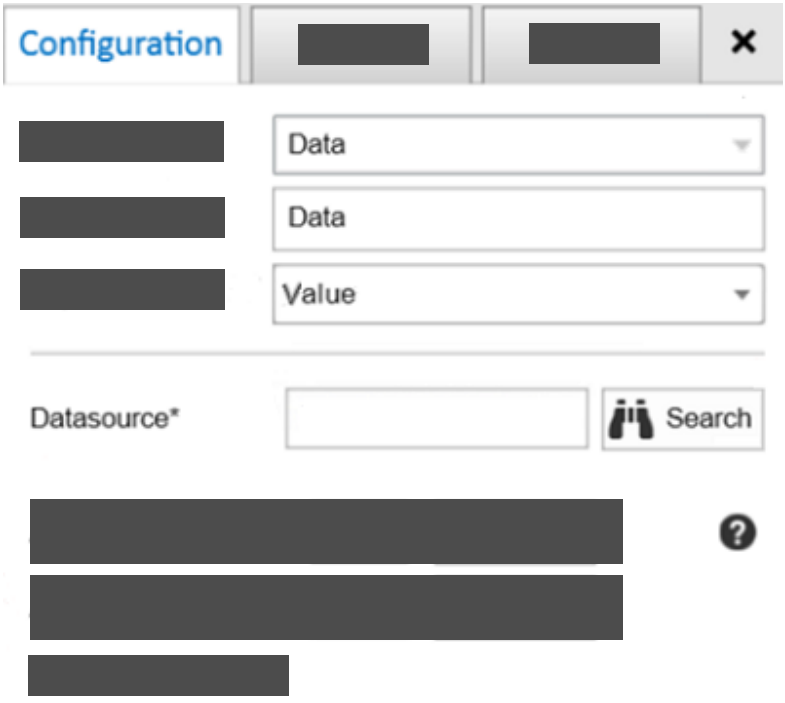
1. Click on the configuration button for the required data object.

Data object configuration



2. In the configuration screen, go to the *Datasource* field and click the Search button.

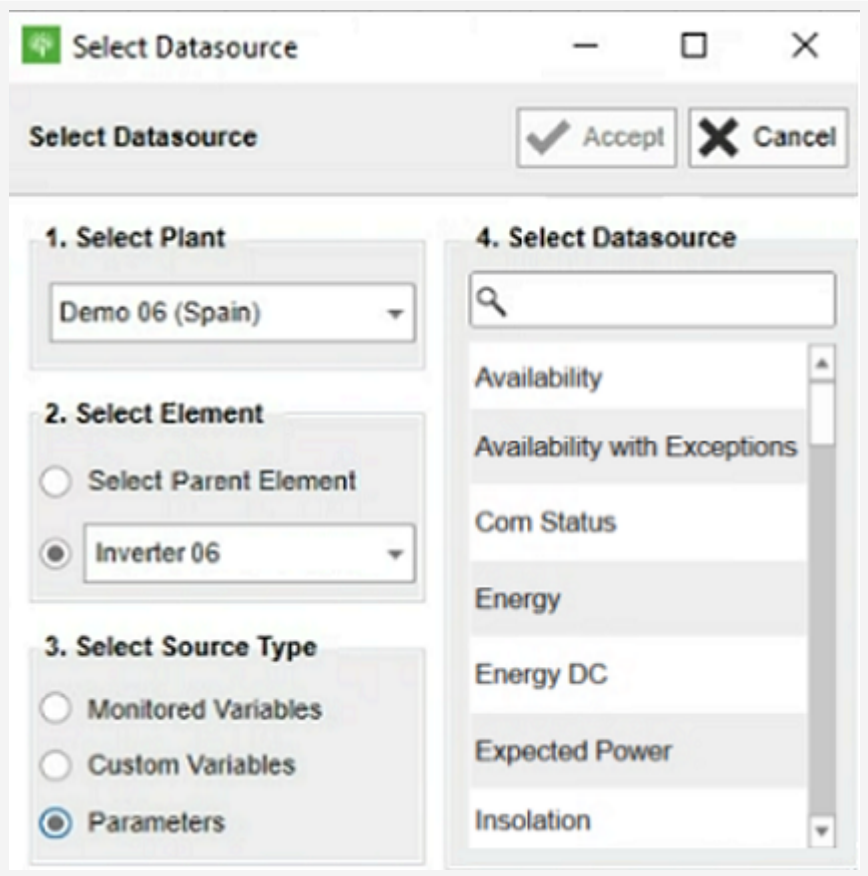
New Datasource field



3. In the Select Datasource window, do the following:

- 2. Select Element: select the element in the plant to which you want to assign parameters.
- 3. Select Source Type: select **Parameters**. The list of available parameters will then be displayed.
- 4. Select Datasource: choose the required parameter.

Select Datasource

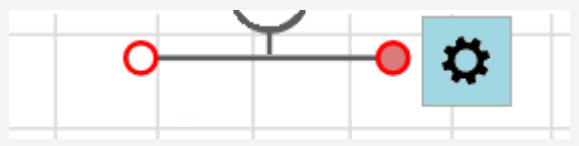


Configuring connection objects in the grid

You can configure connection objects using the corresponding configuration window. To configure connection objects, follow these steps:

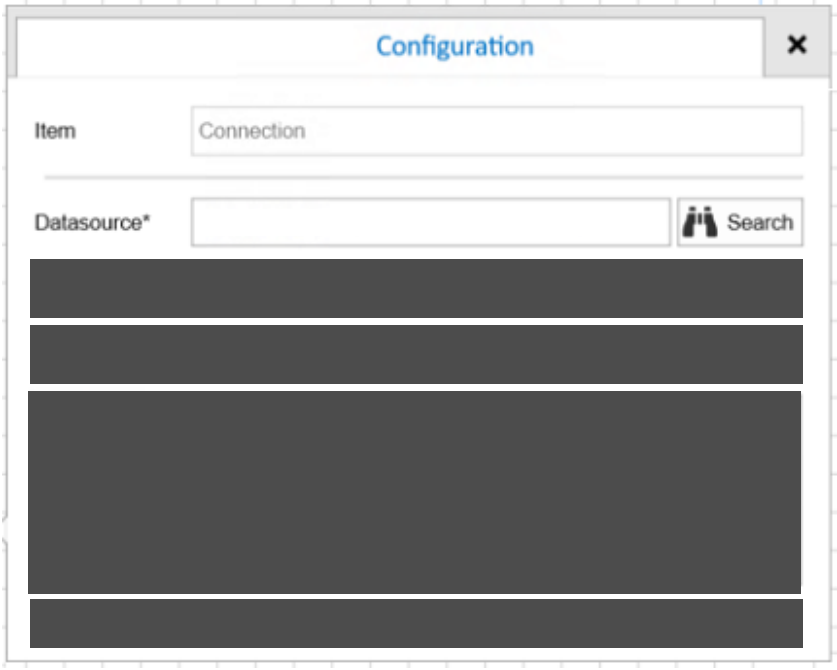
1. Click on the configuration button for the required connection object.

   New Datasource field

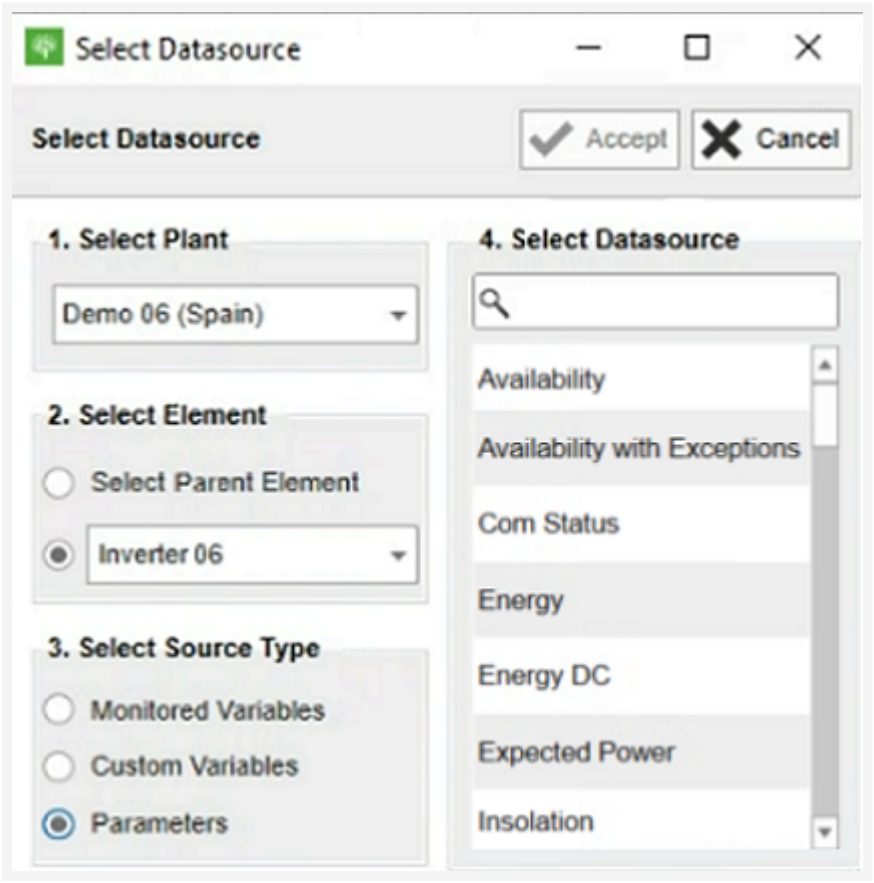
2. In the configuration screen, go to the *Datasource* field and click the Search button.

   New Datasource field

3. In the Select Datasource window, do the following:

Select Datasource



2. Select Element: select the element in the plant to which you want to assign parameters.

3. Select Source Type: select **Parameters**. The list of available parameters will then be displayed.

4. Select Datasource: choose the required parameter.

5

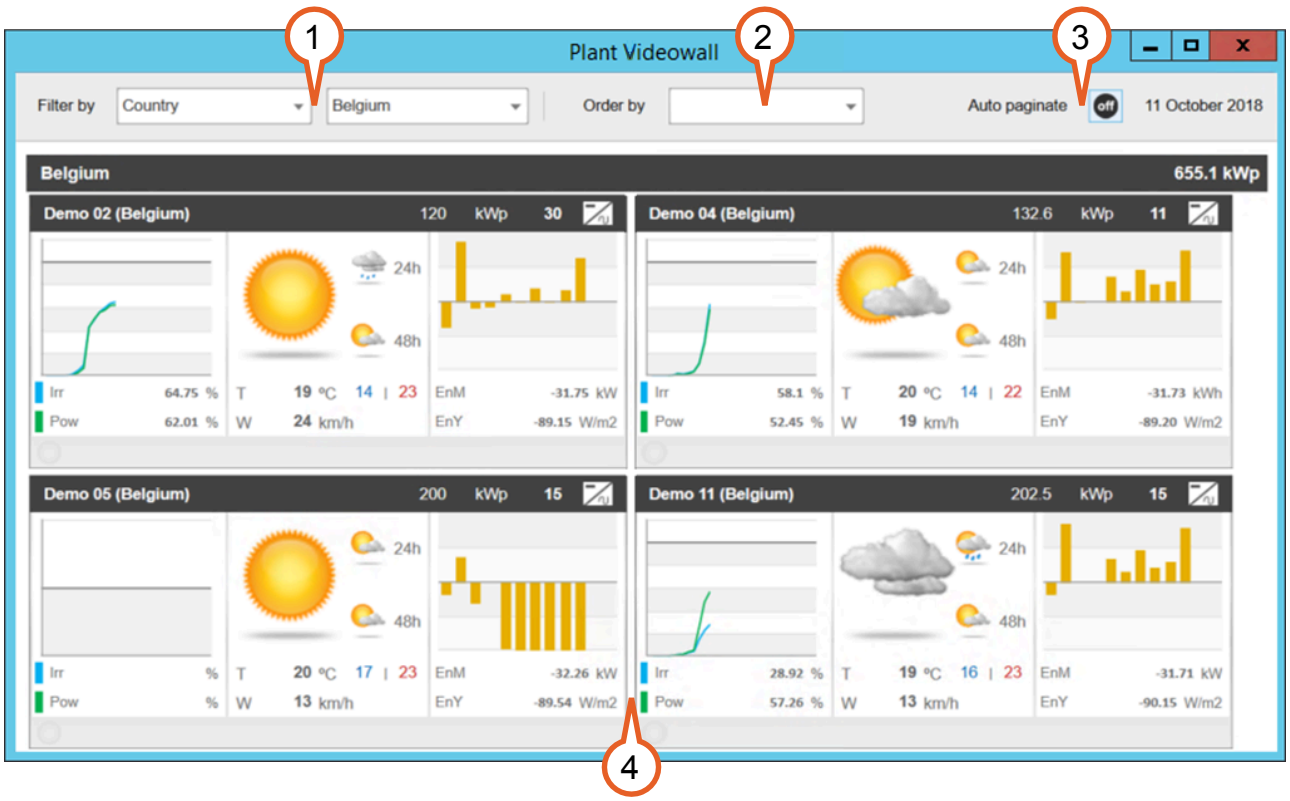
Result

Videowall

The Videowall module offers a general portfolio overview for monitors mounted on walls. Videowall displays the most up-to-date content and automatically adapts to the window size. You can customize the module to display the most relevant productive parameters, the weather conditions, and the accumulated tendency.

To access the Videowall module, click the  icon on the Upper bar, then click the icon.

Videowall

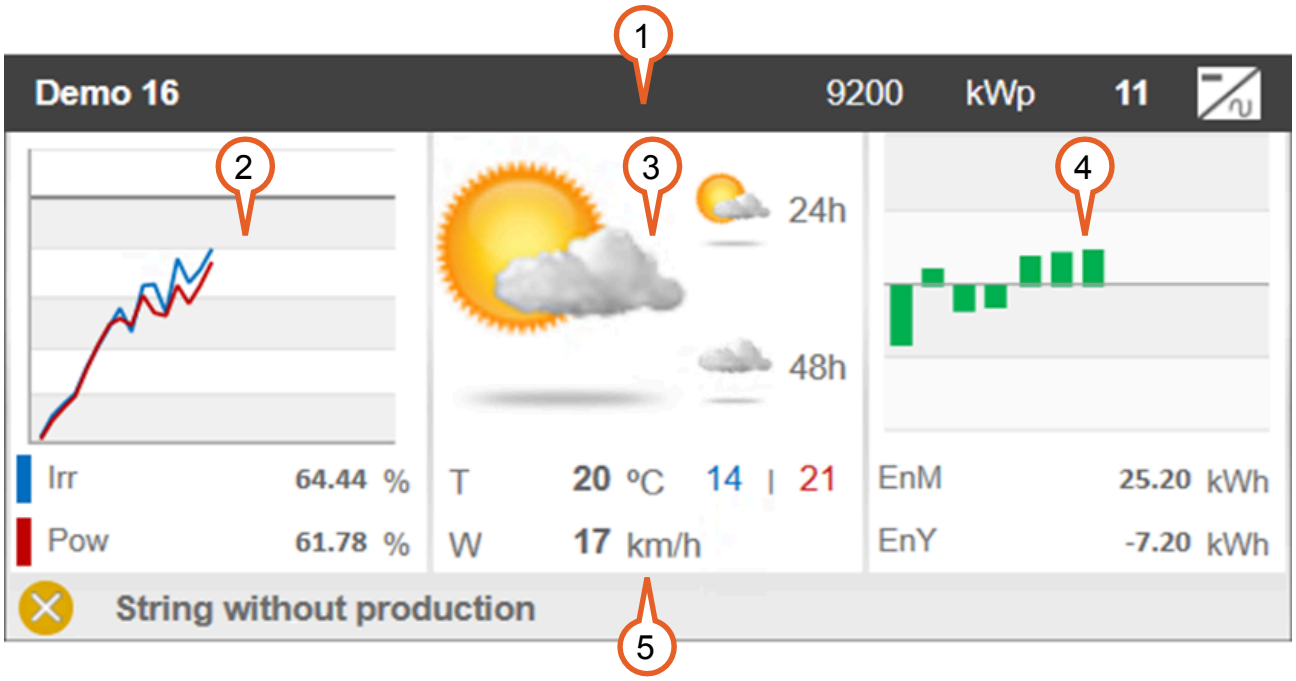


1. **Filter**: click to filter plants by selecting a criterion and a value from the drop-down lists
2. **Order by**: click to sort panels by name or by peak power.
3. **Auto-pagination**: click to switch on or off:
   - **On**: the page scrolls automatically to display all the cards.
   - **Off**: you must scroll the page manually to display cards that do not fit in the window.
4. Cards: display each plant as a card, sorted alphabetically.

Videowall cards

The cards in the Videowall consist of three panels. Each panel provides general information about the plant, instant values, weather conditions, accumulated values, and the most relevant alarm for the plant.

Videowall card



1. Upper ribbon: displays the plant name, the peak power in kWp, and the number of installed solar trackers and inverters.
2. Real-time chart: monitors the evolution of two predefined parameters in real time. For the best graphic output, we recommend using parameters expressed as percentages
3. Weather: displays the current weather and the forecast for the following two days. Displays the current, minimum and maximum temperature, and the wind speed.

   The information is displayed based on the plant coordinates and is retrieved from an external provider, so that you can compare third-party information with the data from the plant sensor.
4. Accumulated values: provides a quick summary of a KPI by comparing a budget value and a real value. Data is provided as a chart and as a value list
5. Lower ribbon: displays the most relevant alarm for the plant, if any.

Outages

Outages allow you to plan the exclusion of a period from the budget calculations. This lets the system account for future production downtimes caused by external events (such as maintenance), issues, or electrical network requests.

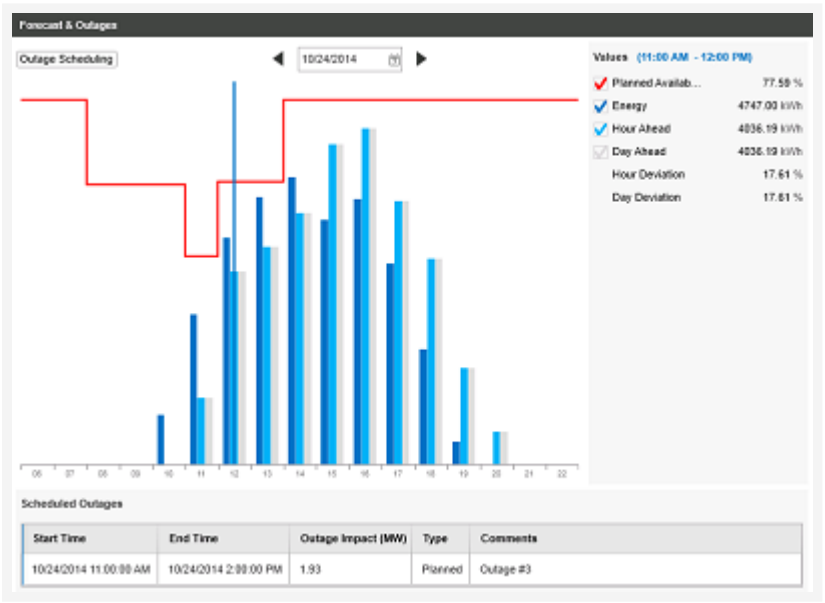
You can schedule an outage in the Forecast & Outages panel of the Plant Dashboard module.

Schedule an outage

To schedule an outage, follow these steps:

1. Open the Plant Dashboard module and expand the Forecast & Outages panel.

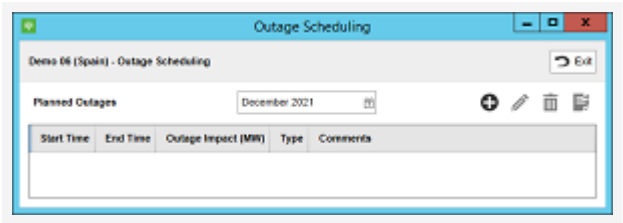
Forecast & Outages panel



2. Click Outage Scheduling.

   Result: The **Outage Scheduling** dialog appears:

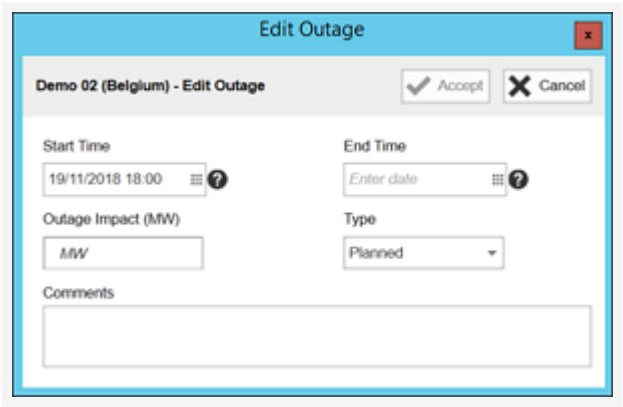
Outage Scheduling dialog



3. Click the ⊕ icon.

Result: The **Edit Outage** dialog appears:

Edit Outage dialog



4. Enter the information for the outage:
   a. **Start time**: select the date and time at which the outage starts.
   b. **End time**: select the date and time at which the outage ends.
   c. *Outage impact*: enter the impact of the outage Megawatts (MW).
   d. Type: select the type of outage from the drop-down list.
      - **Planned**
      - **Unplanned**
      - **Disabled**
   e. Comments: enter any comments you think necessary.
5. Click **Accept**.

Result

The outage is scheduled.

Permissions

The GPM permissions system allows you to restrict the access that different users have to specific entities (parameters, datasources, descriptions and alarms) by applying tags to specific user roles.

For example, you can make "maintenance" alarms and tickets accessible to everyone, while keeping "operations" alarms and tickets accessible only to operations and management (O&R)

users.

You can manage permissions and tags in the Permissions module.

Tags

A tag is an attribute that links specific entities to user roles. Once you apply a tag to an entity, it becomes exclusive: only users with the same tag have access to it.

Manage permissions

Permissions work by using tags to link roles and the entities to which they have access. When you assign a tag to an entity, it becomes accessible only to the user roles which also have the tag assigned to them.

> ⓘ NOTE: All the tasks related to managing permissions take place in the Permissions module.

Create and assign permissions

The basic process to create and assign permissions consists of three steps:

1. Create tag.
2. Add entities to the tag:
    - Alarms
    - Datasources
    - Descriptions
    - Parameters
3. Assign the tag to a user role.

Tag management tasks

There are several tasks to manage tags:

- Create new tags.
- Edit existing tags by modifying the entities in them:
    - Add alarms
    - Add datasources
    - Add descriptions
    - Add parameters
    - Delete entities
- Copy tags to easily create new tags based on existing ones.
- Delete tags.

Access management tasks

There are two main tasks to manage access:

- Assign a tag to a user role.
- Unassign a tag from a user role.

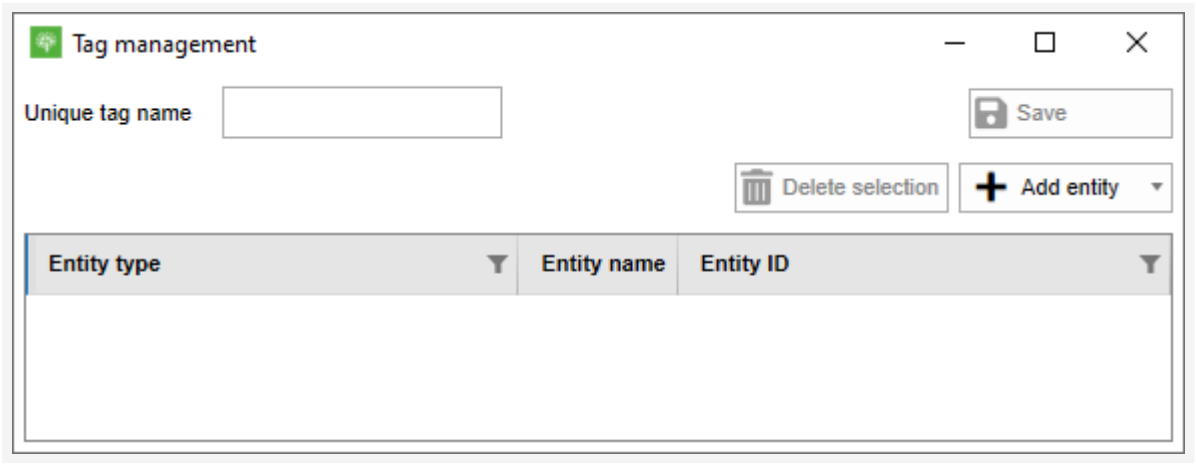
Create tags

To create tags for permissions, follow these steps:

1. In the Manage tags tab of the Permissions module, click **+ Add**.

   Result: The Tag Management dialog appears:

   Tag management dialog

2. In the Tag management dialog, enter a *Unique tag name*.
3. Click **Save**.

Result

The tag is created and you can add entities to it:

- Add alarms
- Add datasources
- Add descriptions
- Add parameters

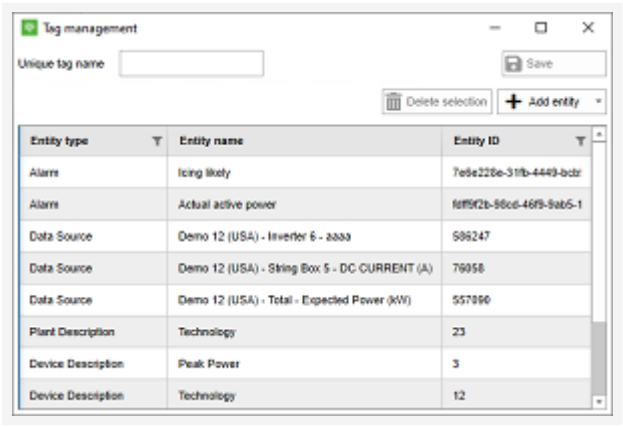
Copy tags

To copy a tag, follow these steps:

1. In the Manage tags tab of the Permissions module, click **Copy**.

   Result: The Tag Management dialog appears:

   ### Tag management dialog

2. Enter a *Unique tag name*.
3. Click **Save**.

Result

The tag is copied and you can edit it:

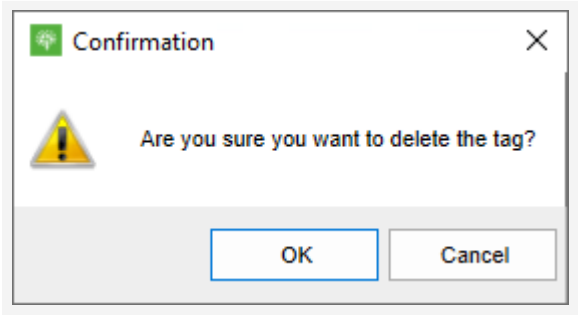
- Delete entities from tags
- Add entities to tags:
  - Alarms
  - Datasources
  - Descriptions
  - Parameters

Delete tags

To delete a tag, follow these steps:

1. In the Manage tags tab of the Permissions module, select the tag and click **Delete**.
   Result: A confirmation message appears:

   Confirmation dialog

2. Click **OK**.

Result

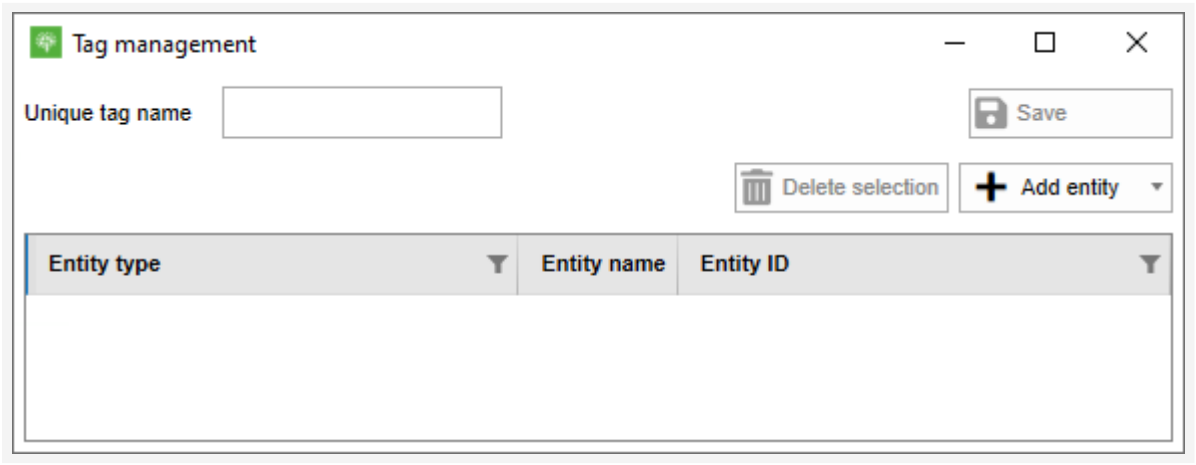
The tag is deleted.

Add alarms to tags

To add alarms to a tag, follow these steps:

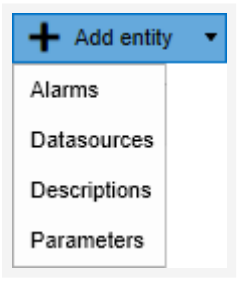
1. In the Manage tags tab of the Permissions module, select a tag and click **Edit**.
   Result: The Tag Management dialog appears:

Tag management dialog



2. Click **+ Add** entity.
   Result: The **Entities** drop-down menu appears.

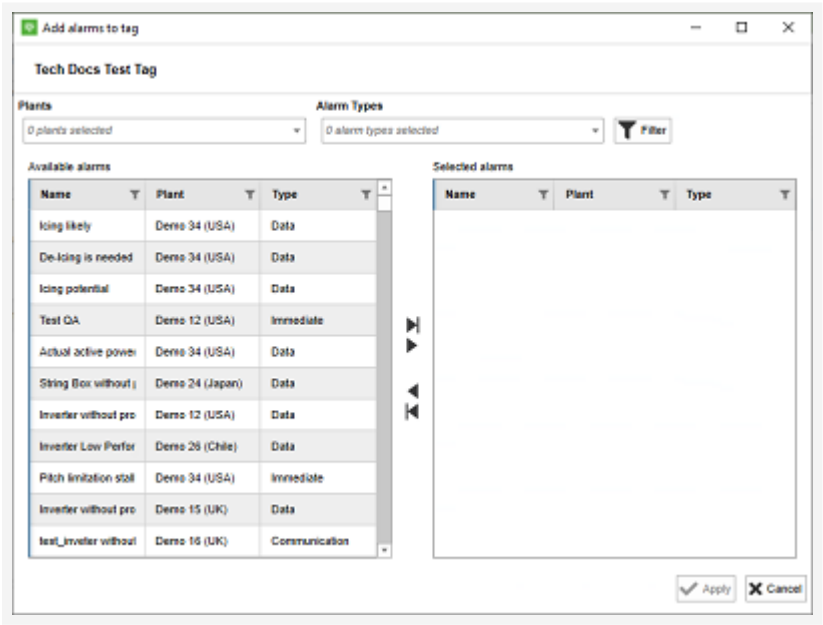
Entities menu



3. Select **Alarms**.

Result: The **Add alarms to tag** dialog appears:

Add alarms to tag



4 In the **Add alarms to tag** dialog, follow these steps:

a (Optional) To refine the list of available armas, click the drop-down menus to select **Plants** and **Alarm types**, then click **Filter**.

Result: The Available alarms panel displays the filtered results.

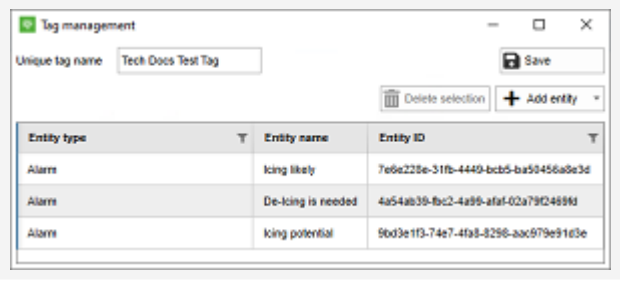
b In the **Available alarms** panel, select the alarms you want to add to the tag and click the ▶ icon. To select all the available alarms, click the ▶| icon.

Result: The alarms move to the **Selected alarms** panel.

a Click ✓ **Apply**.

Result: The Add alarms to tag dialog closes and the Tag management dialog displays the selected alarms.

Tag management dialog



5 In the **Tag management dialog**, click **Save**.

Result

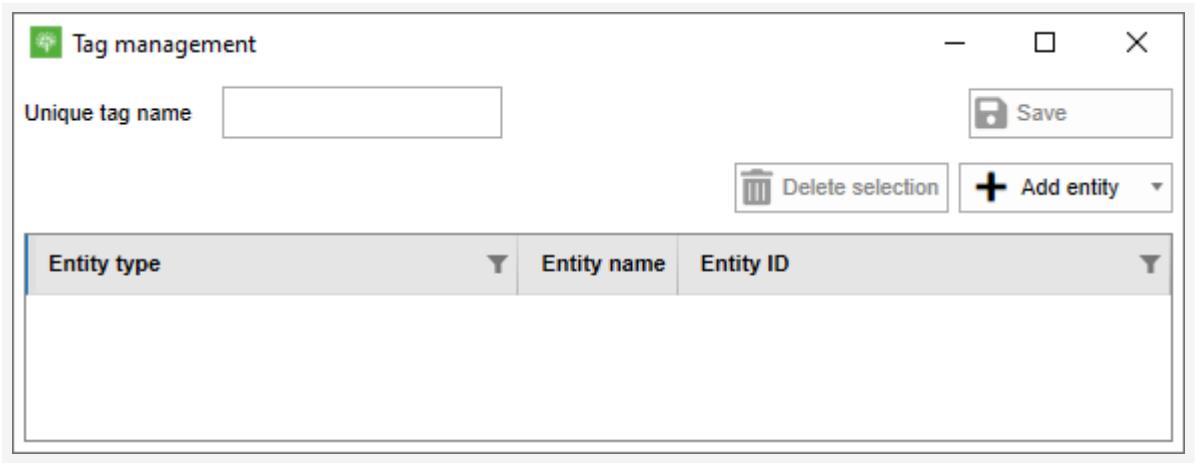
The alarms are added to the tag.

Add datasources to tags

To add datasources to a tag, follow these steps:

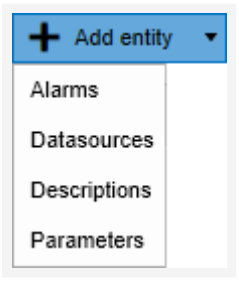
1. In the Manage tags tab of the Permissions module, select a tag and click **Edit**.
   Result: The Tag Management dialog appears:

Tag management dialog



2. Click **+ Add** entity.
   Result: The **Entities** drop-down menu appears.

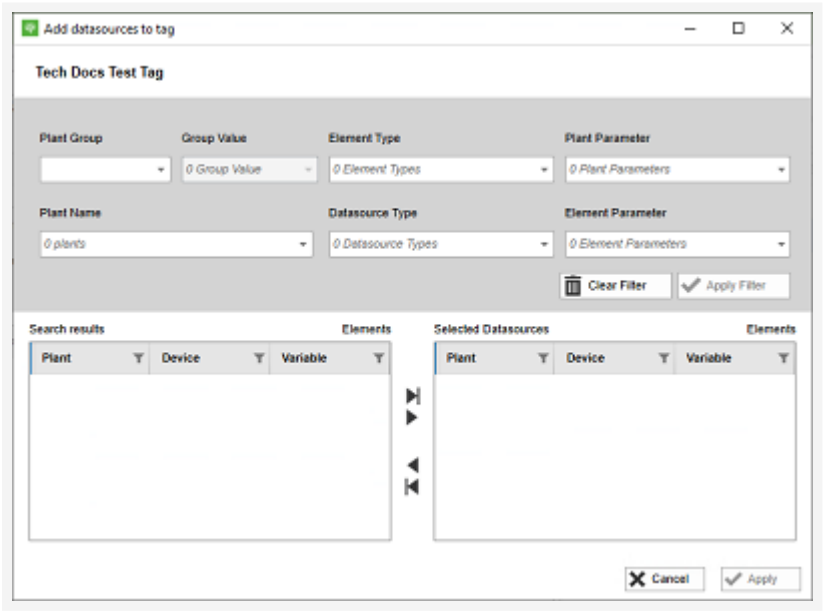
Entities menu



3. Click **Datasources**.

Result: The **Add datasources to tag** dialog appears:

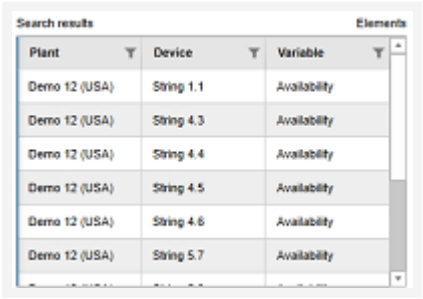
Add datasources to tag dialog



4. Click the **Plant name** drop-down menu and select the plants from which you want to add datasources.
5. (Optional) Apply Advanced filters to refine the list of available datasources.
   a. Select a **Plant group** (for example, **Country**) and at least one Group value (for example, **Belgium**).
   b. Click the **Element type** drop-down menu and select types of element (for example, **Inverter** or **Generator**).
   c. Click the **Plant parameters** drop-down menu and select parameters (for example, **Energy** or **Power**).
   d. Click the **Datasource type** drop-down menu and types of datasources (for example, **Availability** or **Production Ratio**).
   e. Click the **Element parameter** drop-down menu and select element parameters (for example, **Availability** or **Alarm**).
6. Click **Apply filter**.

Result: The **Search results** panel displays the datasources that meet the filtering criteria.

Available datasources



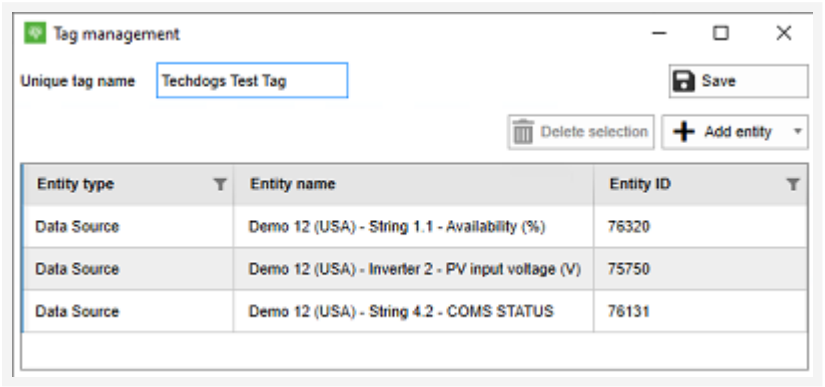
7. In the **Search results** panel, select the datasources you want to add to the tag and click the ▶ icon. To select all the available datasources, click the ▶| icon.

   Result: The datasources move to the **Selected datasources** panel.

8. Click **Apply**.

   Result: The Add datasources to tag dialog closes and the Tag management dialog displays the selected datasources.

Tag management dialog



9. In the **Tag management dialog**, click **Save**.

Result

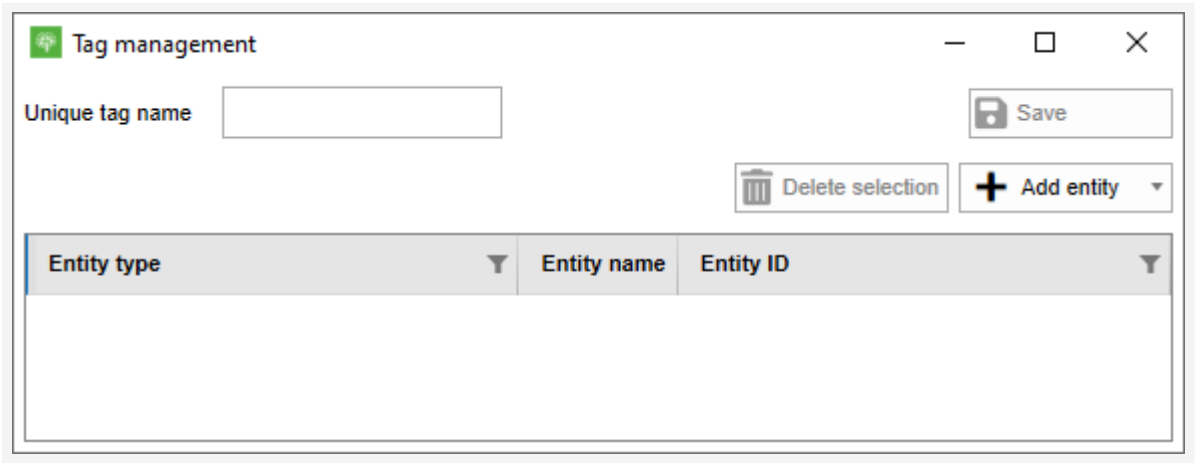
The datasources are added to the tag.

Add descriptions to tags

To add descriptions to tasks, follow these steps:

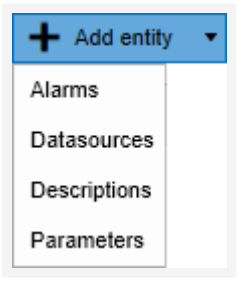
1. In the Manage tags tab of the Permissions module, select a tag and click **Edit**.
   Result: The Tag Management dialog appears:

Tag management dialog



2. Click **+ Add** entity.
   Result: The **Entities** drop-down menu appears.

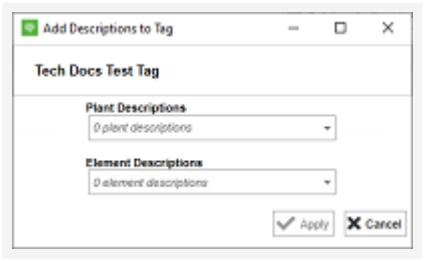
Entities menu



3. Click **Descriptions**.

Result: The **Add descriptions to tag** dialog appears:

Add descriptions to tag dialog



4. In the **Add descriptions to tag** dialog, select the descriptions you want to add to the tag.

> ⓘ NOTE: You can add Plant descriptions, Element descriptions, or both.

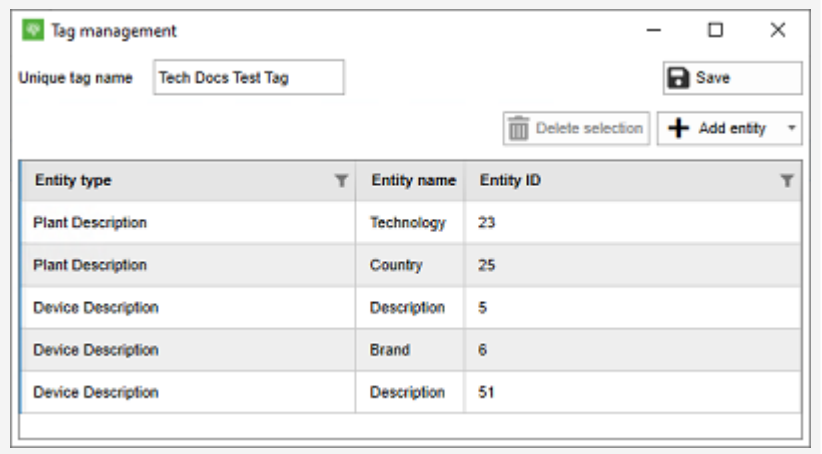
a. Click the **Plant descriptions** drop-down menu and select plant descriptions (for example, **Country** or **Technology**).

b. Click the **Element descriptions** drop-down menu and select element descriptions (for example, **Brand** or **Peak Power**).

c. Click ✓ **Apply**.

Result: The Add descriptions to tag dialog closes and the Tag management dialog displays the selected descriptions.

Tag management dialog



5. Click **Save**.

Result

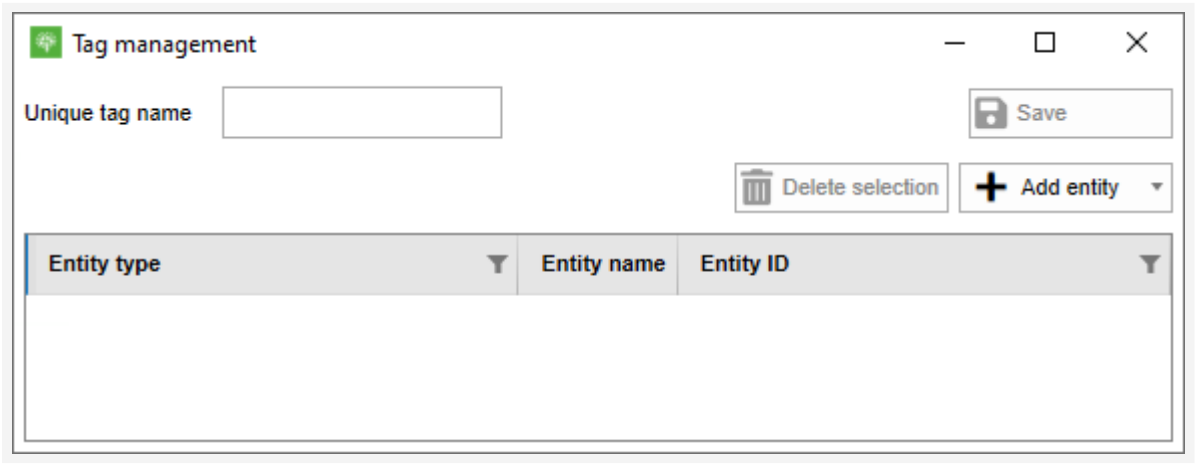
The descriptions are added to the tag.

Add parameters to tags

To add parameters to a tag, follow these steps:

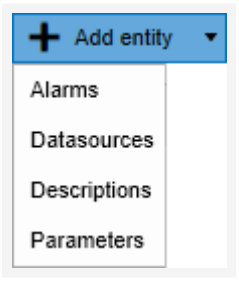
1. In the Manage tags tab of the Permissions module, select a tag and click **Edit**.
   Result: The Tag Management dialog appears:

Tag management dialog



2. Click **+ Add** entity.
   Result: The **Entities** drop-down menu appears.

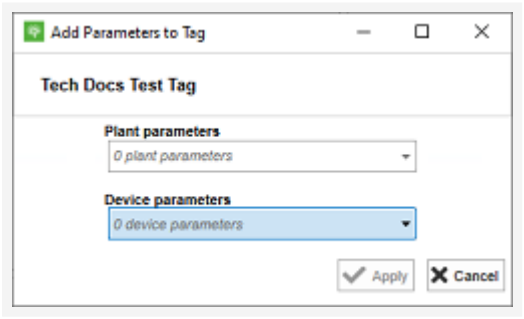
Entities menu



3. Click **Parameters**.

Result: The **Add parameters to tag** dialog appears:

Add parameters to tag dialog



4. In the **Add parameters to tag** dialog, select the descriptions you want to add to the tag.

> ⓘ NOTE: You can add Plant parameters, Element parameters, or both.

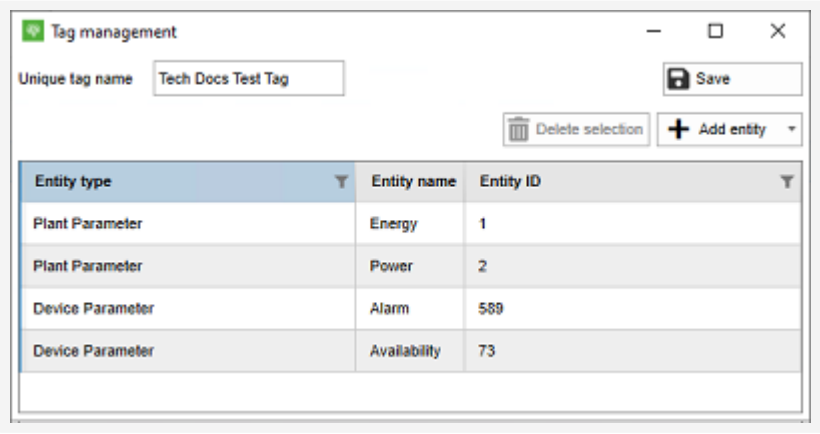
a. Click the **Plant parameters** drop-down menu and select plant parameters (for example, **Energy** or **Power**).

b. Click the **Element parameters** drop-down menu and select element parameters (for example, **Alarm** or **Availability**).

c. Click ✓ **Apply**.

Result: The Add parameters to tag dialog closes and the Tag management dialog displays the selected parameters.

Tag management dialog



5. Click **Save**.

Result

The parameters are added to the tag.

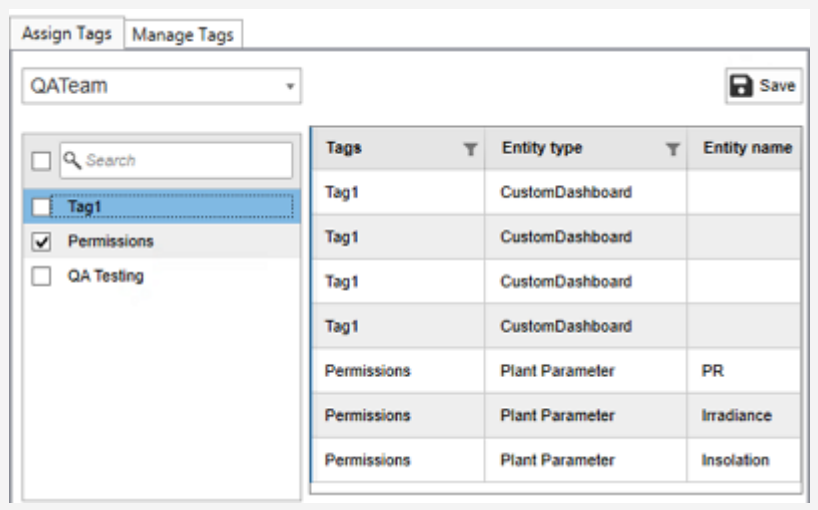
Assign tags to user roles

To assign a tag to a user role, follow these steps:

1. In the **Assign tags** tab of the Permissions module, open the **Select role** drop-down menu and click on the role to which you want to assign a tag.
2. Check the checkbox for the tag you want to assign to the role.

   Result:

   Selected tag



3. Click **Save**.

Result

The tag is assigned to the user role.

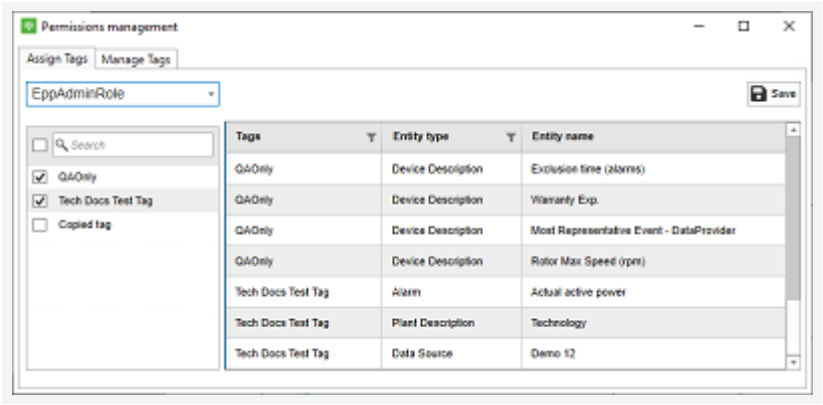
Unassign tags from user roles

To unassign a tag from a user role, follow these steps:

1. In the Assign Tags tab of the Permissions module, click the Roles drop-down menu and select a user role.

   Result: The tags assigned to the role appear:

   Tags and entities

2. Deselect the checkboxes of the tags you want to unassign from the role.
3. Click **Save**.

Result

The tag is unassigned from the user role and those users no longer have access to the entities in the tag.

Plant Data

The GPM system constantly processes and transmits data from your organization's assets to provide you with reliable and up-to-date information for monitoring and in-depth analysis.

You can launch queries to analyze data from plants through three modules:

- Data viewer module
- Linear Chart Viewer
- Scatter Plot Chart

Data Viewer queries

The Data Viewer is a tool that allows you to create queries and analyze the data of your portfolio. The results of the query are arranged in tables that display the values for the selected parameters at specific moments in time. You can save the results of your queries for further use or export them to other views.

You can add queries to the Data Viewer from other areas of the user interface or create queries directly from the Data Viewer module.

Queries consist of two main sections: parameters and time periods. You can select as many parameters as you want, for a single time period. You can also refine the granularity of the data at different levels, from years to minutes. You can also group and aggregate data by different criteria, and apply operations to give it greater complexity.

> ⓘ NOTE: Data granularity affects processing times. For example, selecting 1-minute data for large periods of time may cause the query to take longer to load.

- Create queries
  - Create individual queries

    Create and configure recurring queries
- Edit queries:
  - Change time period
  - Advanced data granularity
  - Advanced parameter configuration

Data granularity

Granularity defines the level of detail for gathering and processing information. In the GPM system, data granularity is defined in units of time (for example, 5-minute intervals).

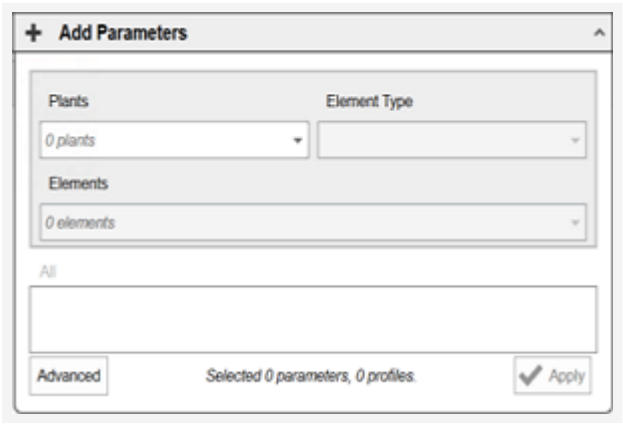
The optimal level of granularity that you select for a data query depends on the kind of analysis you want to perform. For example, a financial analysis of the yearly performance of a plant may require monthly or weekly data, while diagnosing the performance of individual elements to identify potential issues may require gathering data at intervals of 15 minutes over a single day.

Create queries in the Data Viewer module

To create a query in the Data Viewer module, follow these steps:

1. Click **+Add Parameters** to open the parameters selector.

   Parameters selector

   a. Open the **Plants** drop-down menu and select the plants for which you want to see data.

   b. Open the **Element Type** drop-down menu and select the type of element yo want to include in the query (for example, **Inverter**).

   c. Open the **Elements** drop-down menu and select the specific elements you want to include in the query.

   OR: select the checkbox next to the *Search* bar to include all available elements.

   d. In the **Main Parameters** section, select the parameters you want to include in the query (for example, **Insolation**, **Irradiance** and **Energy**).

   OR: Select the **All** tab to select from a list of all the available parameters.

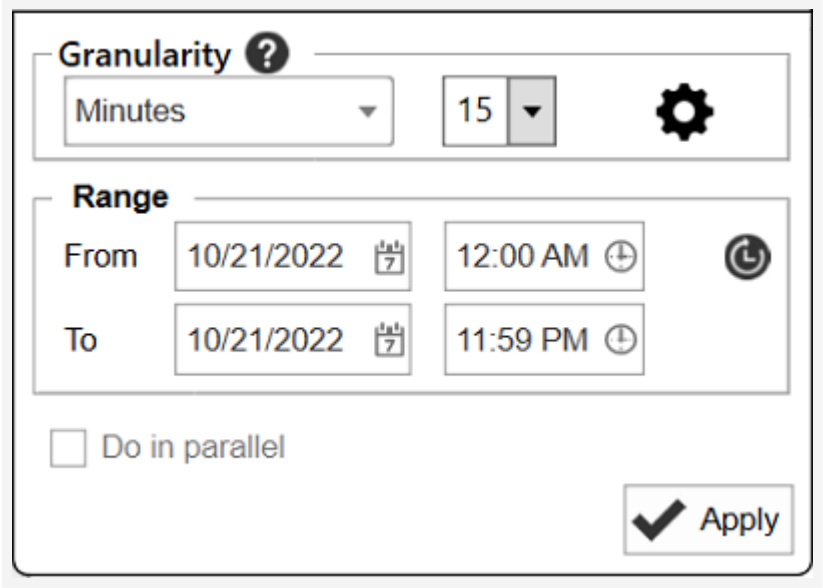
   > ⓘ NOTE: To work with advanced datasources, see the in-depth instructions to Add advanced parameters to queries.

   e. Click ✓ **Apply**.

   f. Click the ^ icon to close the Parameters selector.

2. Click **Period** to open the Time Period and Granularity selector:

Time Period and Granularity selector



a. In the **Granularity** section, open the Grouping drop-down menu to select a grouping method (for example, **Minutes**), then open the Units drop-down menu and specify a value for (for example, **15**).

> ⓘ NOTE: For advanced granularity and grouping configurations, see the in depth instructions to Define advanced data granularity for queries.

b. In the **From** section, select the starting date and time for the query period.

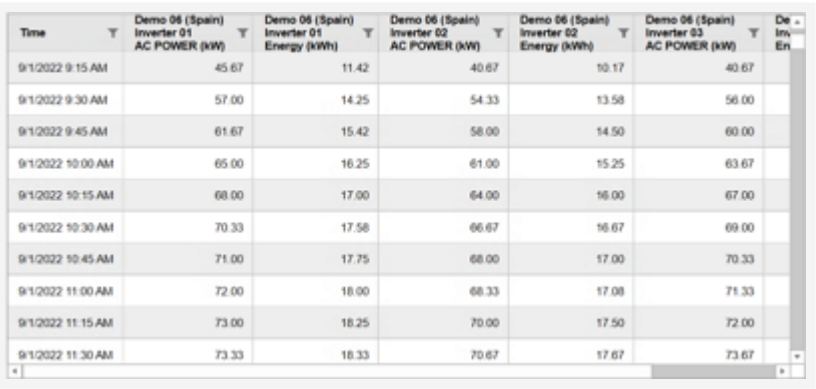
c. In the **To** section, select the end date and time for the query period.

> REMEMBER: For optimal performance, consider the scope and the level of detail you want for the query. Large ranges with high granularity (for example, 15-minute data for a period of one month) take longer to process.

d. Click ✓ **Apply** to launch the query.

Result

Query results



Time	Demo 06 (Spain) Inverter 01 AC POWER (kW)	Demo 06 (Spain) Inverter 01 Energy (kWh)	Demo 06 (Spain) Inverter 02 AC POWER (kW)	Demo 06 (Spain) Inverter 02 Energy (kWh)	Demo 06 (Spain) Inverter 03 AC POWER (kW)	De Inv En
9/1/2022 9:15 AM	45.67	11.42	40.67	10.17	40.67	
9/1/2022 9:30 AM	57.00	14.25	54.33	13.58	56.00	
9/1/2022 9:45 AM	61.67	15.42	58.00	14.50	60.00	
9/1/2022 10:00 AM	65.00	16.25	61.00	15.25	63.67	
9/1/2022 10:15 AM	68.00	17.00	64.00	16.00	67.00	
9/1/2022 10:30 AM	70.33	17.58	66.67	16.67	69.00	
9/1/2022 10:45 AM	71.00	17.75	68.00	17.00	70.33	
9/1/2022 11:00 AM	72.00	18.00	68.33	17.08	71.33	
9/1/2022 11:15 AM	73.00	18.25	70.00	17.50	72.00	
9/1/2022 11:30 AM	73.33	18.33	70.67	17.67	73.67	

The query results appear in the table:

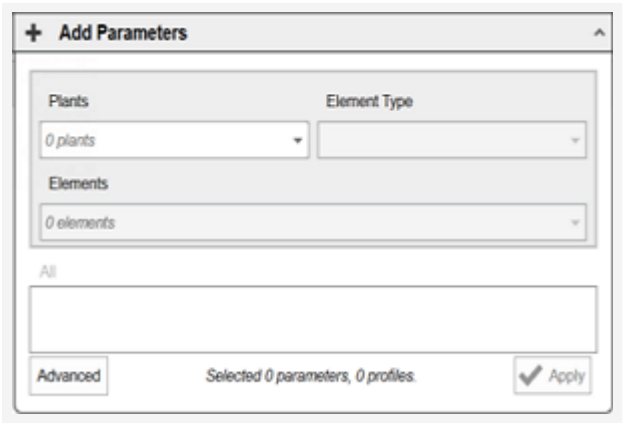
To make this a recurring query, see the instructions to Create recurring queries.

Create recurring queries

To create a recurring query in the Data Viewer module, follow these steps:

1. Click **+Add Parameters** to open the parameters selector.

   Parameters selector

   a. Open the **Plants** drop-down menu and select the plants for which you want to see data.

   b. Open the **Element Type** drop-down menu and select the type of element yo want to include in the query (for example, **Inverter**).

   c. Open the **Elements** drop-down menu and select the specific elements you want to include in the query.

   OR: select the checkbox next to the *Search* bar to include all available elements.

   d. In the **Main Parameters** section, select the parameters you want to include in the query (for example, **Insolation**, **Irradiance** and **Energy**).

   OR: Select the **All** tab to select from a list of all the available parameters.

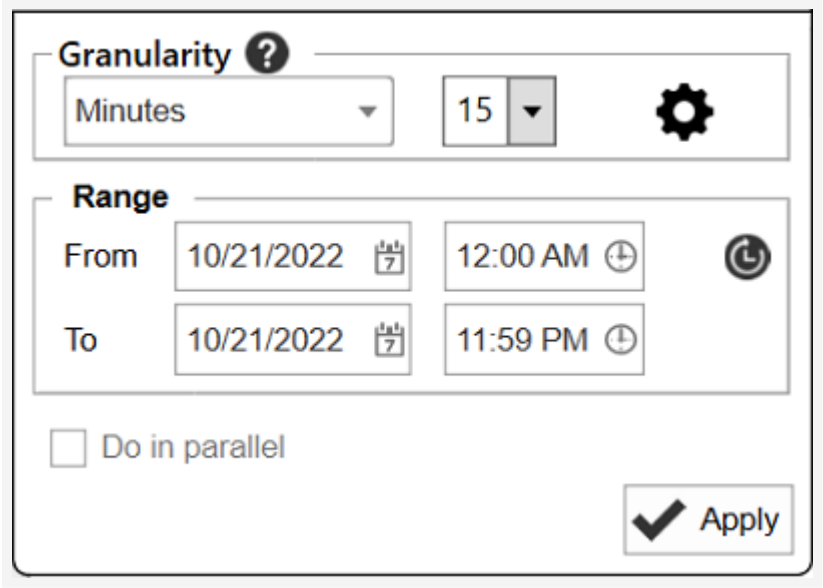
   > ⓘ NOTE: To work with advanced datasources, see the in-depth instructions to Add advanced parameters to queries.

   e. Click ✓ **Apply**.

   f. Click the ^ icon to close the Parameters selector.

2. Click **Period** to open the Time Period and Granularity selector:

Time Period and Granularity selector



a. In the **Granularity** section, open the Grouping drop-down menu to select a grouping method (for example, **Minutes**), then open the Units drop-down menu and specify a value for (for example, **15**).

> ⓘ NOTE: For advanced granularity and grouping configurations, see the in depth instructions to Define advanced data granularity for queries.

b. In the **From** section, select the starting date and time for the query period.

c. In the **To** section, select the end date and time for the query period.

> REMEMBER: For optimal performance, consider the scope and the level of detail you want for the query. Large ranges with high granularity (for example, 15-minute data for a period of one month) take longer to process.

d. Click ✓ **Apply** to launch the query.

3. Click **Save** to open the Save Custom Query window:

a. In the *Save as* field, enter a name for the query.

b. Open the *Frequency* drop-down menu and specify how often you want the system to launch the query (for example, **Weekly**).

c. (Optional) In the *Notes* field, enter any additional information you want to include.

d. Click **Save.**

Result

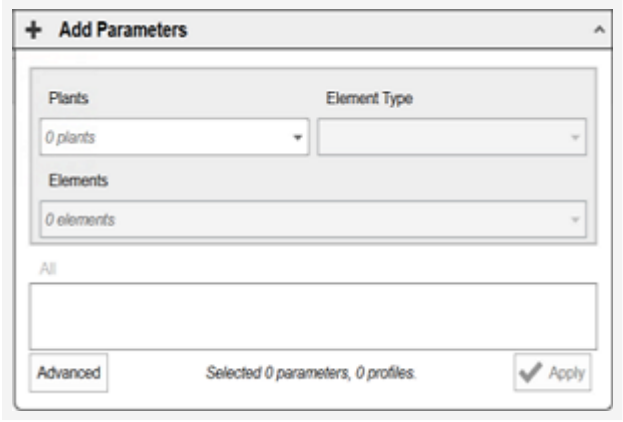
The query is saved and will be launched with the specified frequency.

Add advanced parameters to queries

To add advanced parameters to a query in the Data Viewer module, follow these steps:

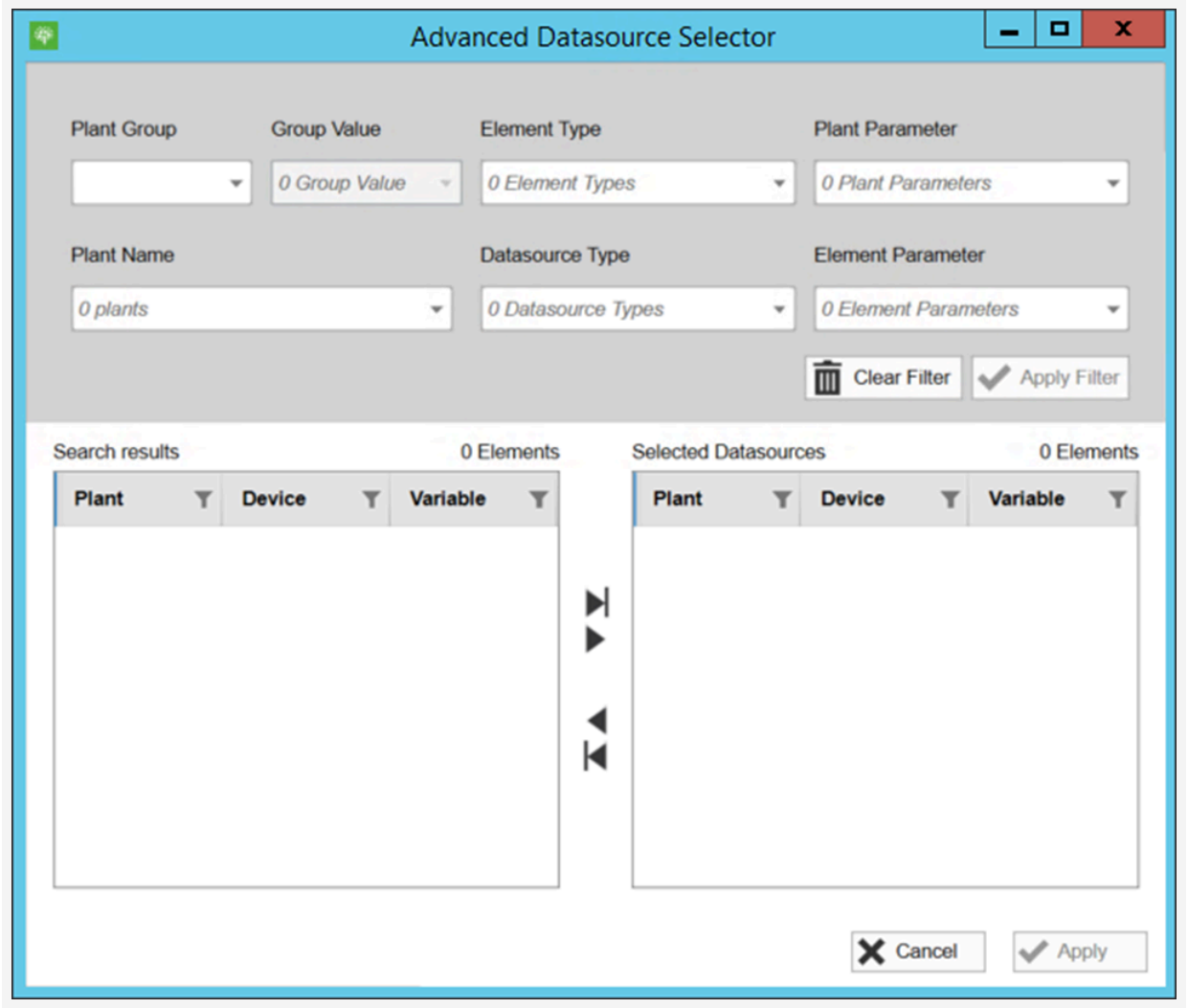
1. Click **+Add Parameters** to open the parameters selector.

   Parameters selector

2. Click **Advanced** to open the Advanced Datasource selector:

Advanced Datasource selector



3. In the **Plant Group** drop-down menu, select the group to which the plant belongs.
4. In the **Group Value** drop-down menu, select one or more values.
5. In the **Element Type** drop-down menu, select one or more types of element.
6. In the **Plant Parameter** drop-down menu, select one or more parameters.
7. In the **Plant Name** drop-down menu, select one or more plants.
8. In the **Datasource Type** drop-down menu, select one or more types of datasource.
9. In the **Element Parameter** drop-down menu, select one or more parameters.
10. Click **Apply filter**.

> ⓘ NOTE: You can clear a filter that was already applied by clicking **Clear Filter**.

11. In the **Search Results** panel, click to select the parameters, then click the ▶ icon to move them to the **Selected Datasources** panel.
12. Click **Apply**.

Result

The parameters are added to the query.

Define advanced data granularity for queries

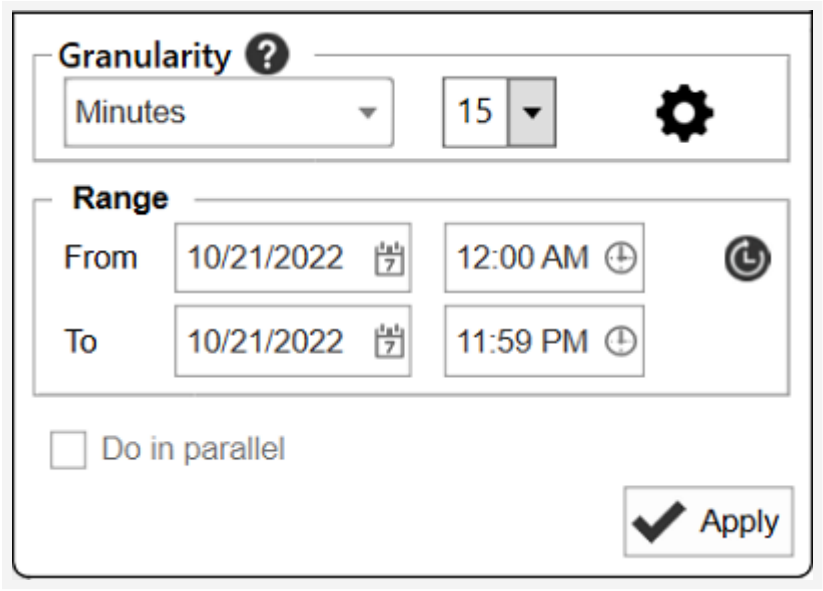
Before you begin

The query must include parameters before you can define the granularity.

To define advanced granularity options for a query in the Data Viewer module, follow these steps:

1. Click **Period** to open the Time Period and Granularity selector:

Time Period and Granularity selector



2. In the **Range** section, define the time period covered by the query:
   a. In the **From** section, select the starting date and time for the query period.
   b. In the **To** section, select the end date and time for the query period.

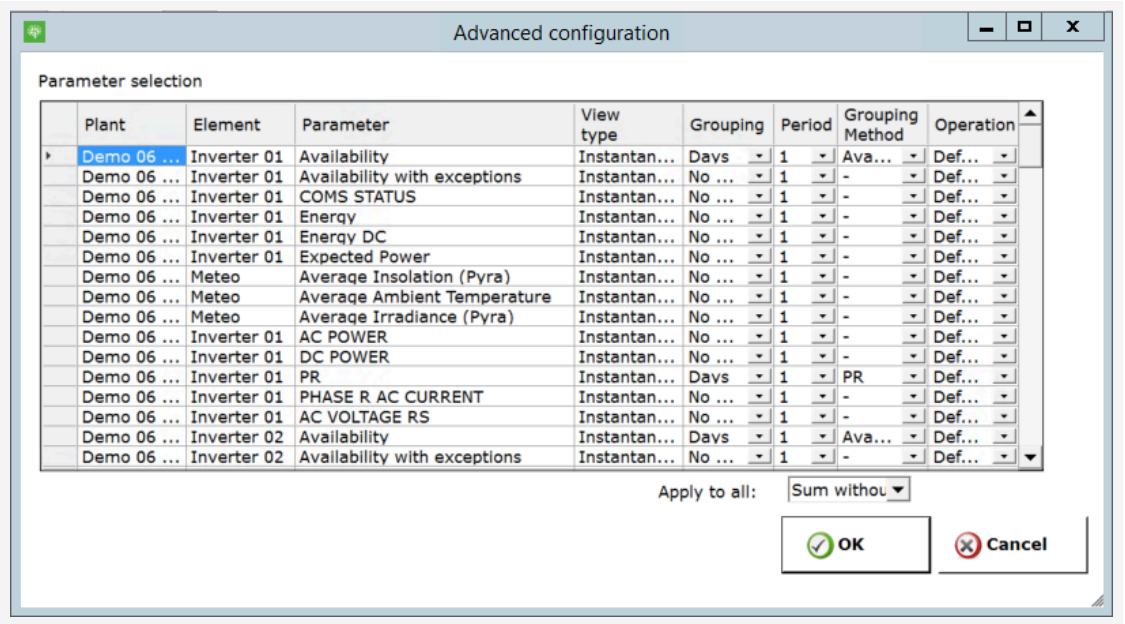
> REMEMBER: For optimal performance, consider the scope and the level of detail you want for the query. Largee ranges of time with high levels of detail (for example, 5-minute data for a period of one month) take longer to process.

3. In the Granularity section, open the **Grouping** drop-down menu to select a grouping method (for example, **Minutes**), then open the **Units** drop-down menu and specify a value for (for example, **15**).

ⓘ NOTE: For advanced granularity and grouping configurations, see the in depth instructions to Define advanced data granularity for queries.

a Click the icon to open the **Advanced Configuration** window:

Advanced Configuration



b In the **Grouping** column, open the drop-down menu to select how to aggregate datapoints (for example, **Minutes**).

c In the **Period** column, open the drop-down menu to define a value for the grouping method (for example, **5**)

d In the **Grouping method** column, select the operation you want to apply to the data (for example, **Average**).

e Click **OK**.

Result

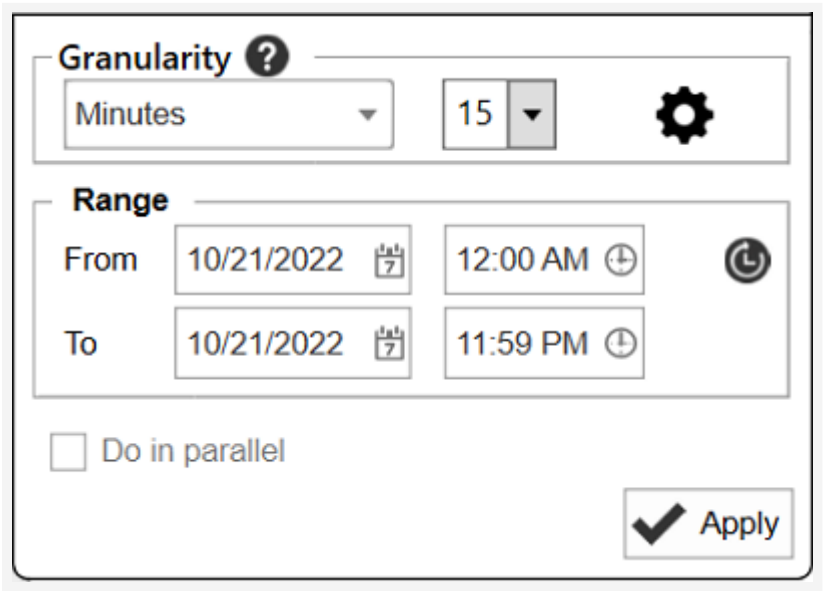
The system applies the changes and runs the query.

Change the time period for a query

To change the time period of a query in the Data Viewer module, follow these steps:

1. Click **Period** to open the Time Period and Granularity selector:

   Time Period and Granularity selector

2. In the **From** section, select the starting date and time for the query period.
3. In the **To** section, select the end date and time for the query period.

   > REMEMBER: For optimal performance, consider the scope and the level of detail you want for the query. Large ranges with high granularity (for example, 15-minute data for a period of one month) take longer to process.

4. Click ✓ **Apply** to launch the query.

Result

The system launches the query to cover the time period you defined.

Reports

Reports are documents that allow you to process and analyze data from your portfolio, export it

to a Microsoft Excel file format, and share data with other users and third parties, such as clients. You can perform tasks related to reports in the Reports module.

You can create individual reports manually, or set up an automatic process to create them at regular intervals and automatically send them to multiple recipients. It is also possible to send reports to Microsoft SharePoint automatically upon creation. For more information, see the Reporting Tasks section.

> ⓘ NOTE: In case of a temporary loss of internet connection, queued reports are automatically sent to
>
> Microsoft SharePoint when the connection is re-established.

Report templates

Templates provide the framework for the information that must be included in a report. The GPM Template Report is the most customizable template, because it allows you to include any data type in your report.

> ⓘ NOTE: The descriptions of tasks to create reports all use the GPM Template Report as an example. There may be some slight variations when working with your custom templates, depending on your configuration.

Reports module

To access the Reports module, click the  icon on the Upper Bar.

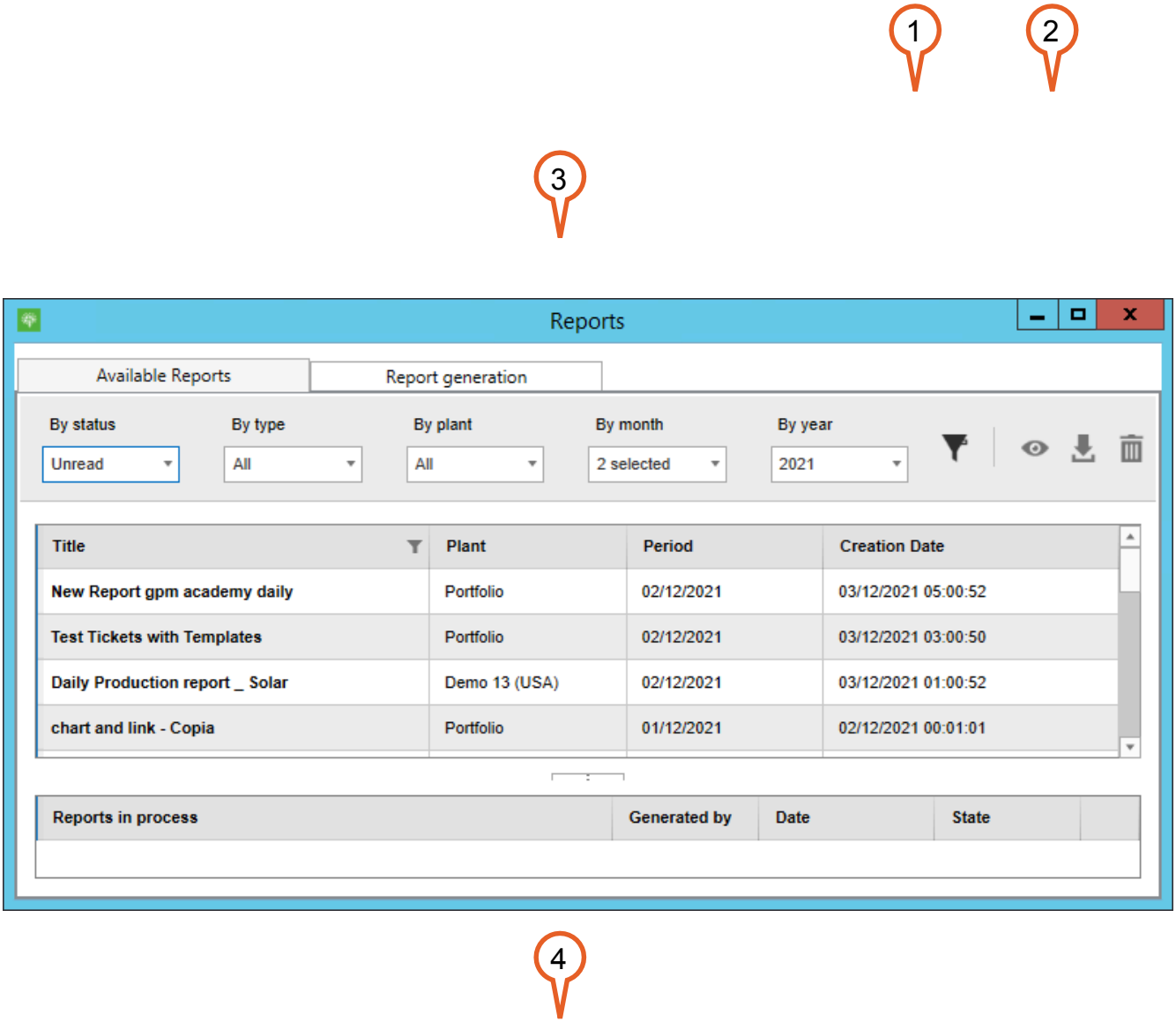
The interface of the Reports module consists of two tabs:

- **Available Reports**: access existing reports.
- **Report Generation**: create new reports.

Available Reports

This tab allows you to access existing reports. By default, the list displays unread reports from the current year and the most recent reports.

Available Reports tab



1. **Template filters**: click to select the filtering criteria from the drop-down lists and

click the icon to display reports that match the criteria on the list.

2. **Action buttons**: click to perform actions on the selected reports:
   - **Open** the selected report. Each report you select opens in a separate window.
   - **Download** the selected report. Each report you select opens a separate download dialog.
   - **Delete** the selected report.
3. Reports list: displays available reports. Select one or more reports to perform actions on them.
   Click a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers. click the icon on the **Title** column header for advanced filtering. For more information, see Advanced Filters.
4. Reports in process: displays reports that are being generated. You can cancel the generation fo a report by clicking the icon on the right side of the list.

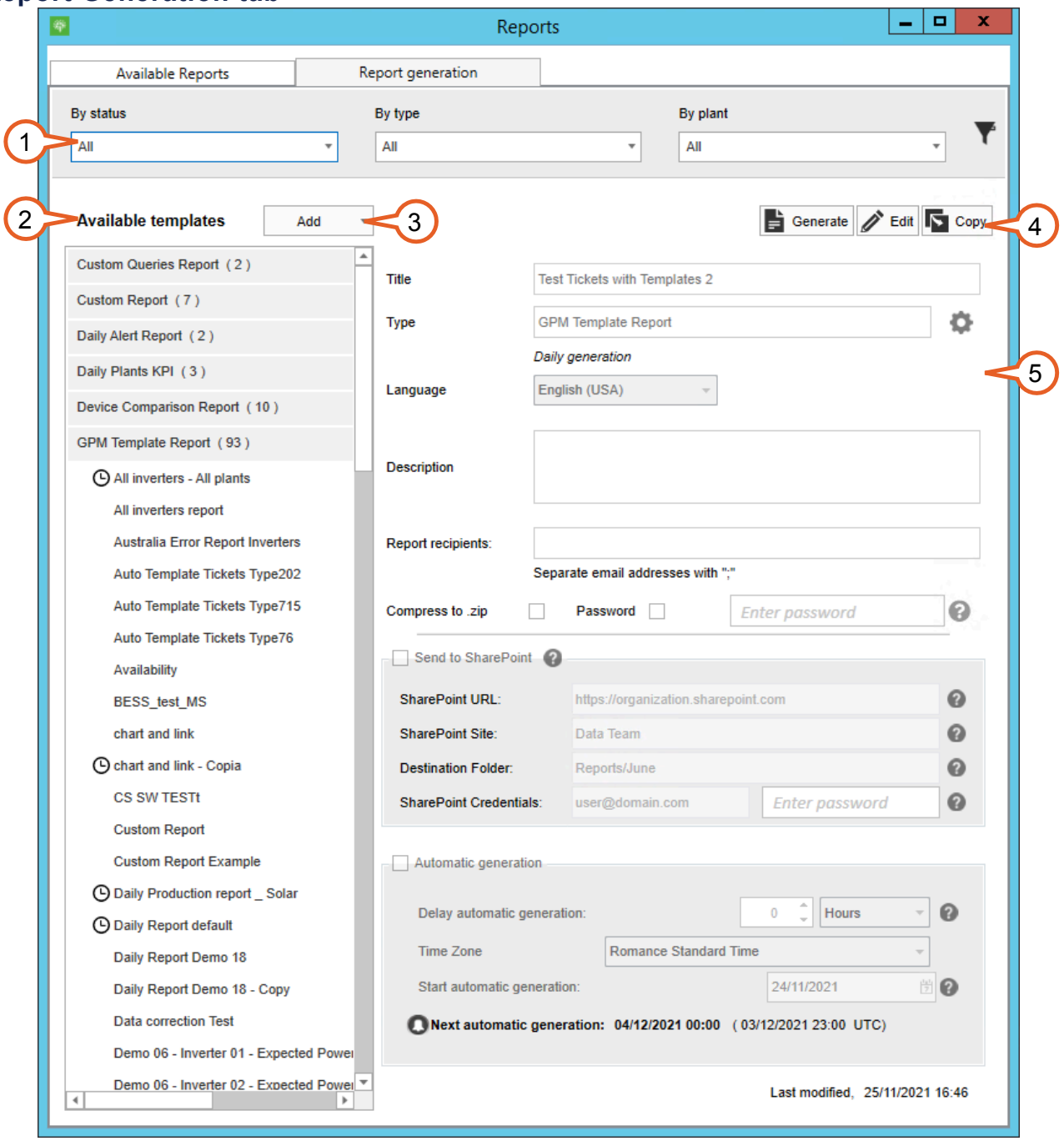
   Click on a column header to sort the table by the values of that column. Rearrange columns by dragging and dropping the headers.

Report Generation

On this tab, you can generate new reports, schedule automatic report generation, and edit existing reports. You can create new reports based on templates that are configured for you.

When you save your report, it becomes available in the **Available templates** panel in the section of the template on which it is based.

Report Generation tab



1. **Template filters**: click to select the filtering criteria from the drop-down lists and click the icon to display reports that match the criteria on the **Available**

templates panel.

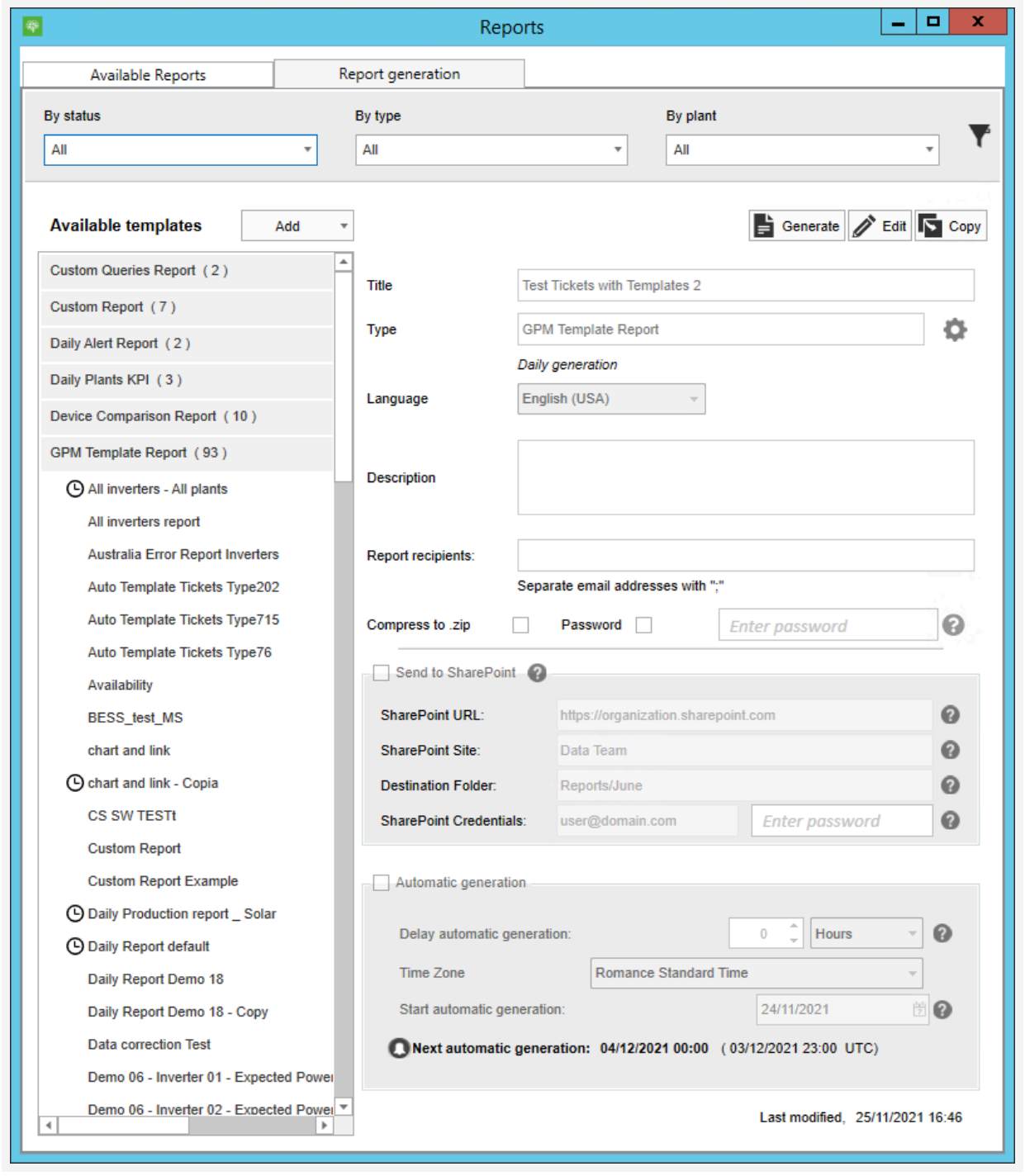
2. Available templates: displays the available reports, organized by report type. Click on a template to edit it.
3. **Add template**: click to select a template from the drop-down list. For further information, see Create Reports using the GPM template.
4. **Action buttons**: click to perform actions on templates or reports:
   - **Generate** one of the selected reports manually. This button is only available for saved templates.
   - **Enable editing** for a selected report. This button is only available for saved templates.
   - **Create a copy** of the selected report. This button is only available for saved templates
   - **Save** your changes. This button is only available for unsaved reports.
   - **Cancel** your changes. This button is only available for saved reports.
   - **Delete** the selected report. This button is only available for saved reports.
5. Report settings: configure the settings for the report. This template becomes available when you add new report or when you are editing one of the available templates.

Create reports using the GPM template

To create a report using the GPM template, follow these steps:

1. In the **Reports** module, go to the **Report Generation tab**:

Report Generation tab



2. In the **Available Templates** panel, click **Add** and select **GPM Template Report**.
3. 3. In the report details area, enter the report information in these fields:
   a. *Title*: enter a title for the report.
   b. *Type*: this field is automatically filled with the template that you choose. To customize the template, see step 4 below.

c **Language**: click to select a culture for the report from the drop-down list.

> ⓘ NOTE: The culture you select affects the report language, decimal sign, and date format. It is possible to define the date format manually by customizing the report template (see step 4 below).

d *Description*: enter a description for the report.

e (Optional) *Report recipients*: if you want the report to be sent via email, enter the recipient's email address here. Multiple addresses must be comma-separated.

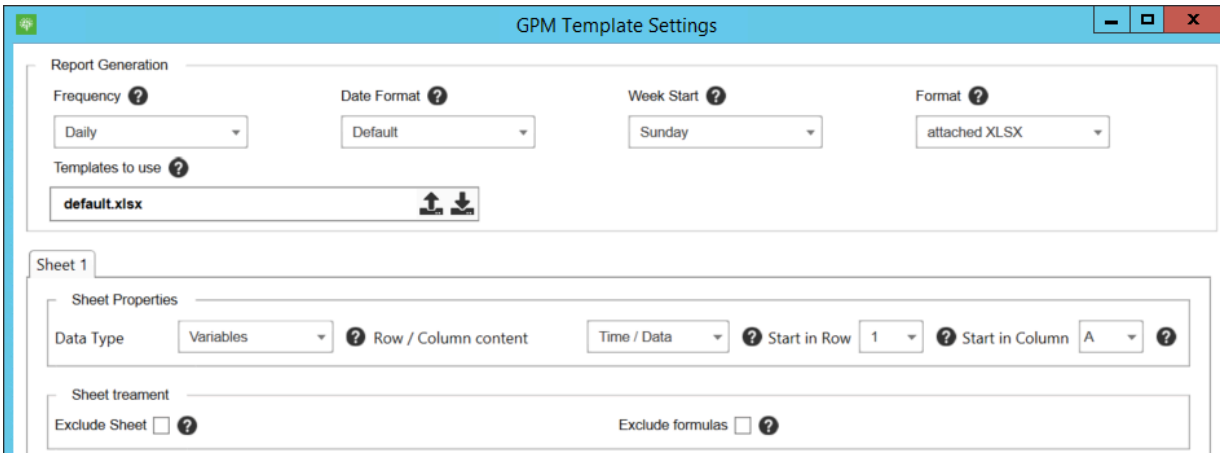
f **Compress to zip**: select this option to compress the report as a ZIP file. If your administrator set a lower size limit for compression, reports will be compressed only when they exceed that size.

g **Password**: select this option and enter a password to encrypt the report with a password.

> ⓘ NOTE: If you select this option, the report is compressed as a ZIP file even if you did not select the previous option.

4 (Optional) To customize the report template, click the ⚙ icon to open the **GPM Template Settings** dialog.

GPM Template Settings



The report can have multiple sheets. Each sheet can only have one type of object, such as variables, tickets or alarms. If you want to include different object types, you must add one sheet for each type.

Click each field to define it:

a **Frequency**: select how often to generate the report generation from the drop-down list.

b **Date format**: the format for dates in the report. By default, the date format matches the language and culture you defined for the report.

c **Week start**: select the first day of the week for reports.

d **Format**: select the format of the file that will be sent to the recipient.

e **Templates to use**: this field is filled automatically with the default Microsoft Excel file for the template. If you want to modify the default template, click the ⬇ icon to download it and then click the ✓ icon to upload the edited version.

f **Data type**: select the type of data for the current sheet.

> ⓘ NOTE: The type you select affects the options available in the Data Selection section (see Sstep 5)

g **Row/Column content**: select the type of data to include in the table. Selecting **Time / Data** places the time stamp in the column before the data. Selecting **Data / Time** places the time stamp in the column after the data.

h **Start in row**: select the row on which the data must start.

i **Start in column**: select the column on which the data must start.

j **Exclude sheet**: select the checkbox to exclude the sheet from the report file.

> ⓘ NOTE: The data contained in the excluded sheets is still used for calculation purposes.

k **Exclude formulas**: select the checkbox to exclude the formulas from the report file.

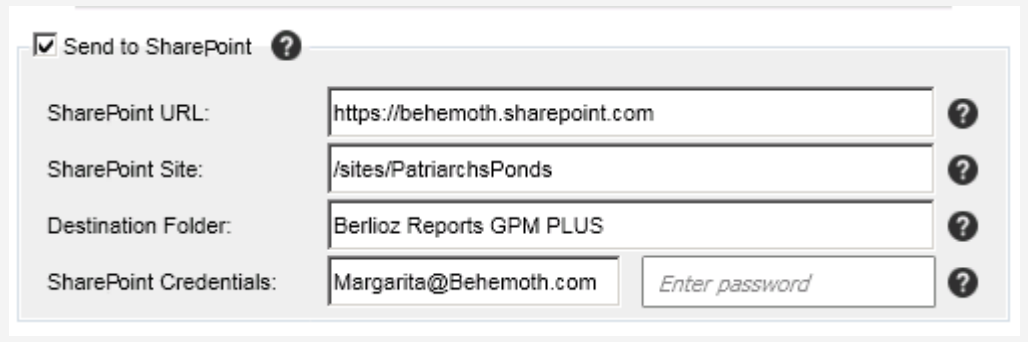
5 Select the data to include in the report and click 💾 **Save**.

The configurations change depending on the type of data you selected step 4f (above). Follow the link below to see the instructions for the type you selected:

- Alarms
- Element Descriptions
- Events
- Exceptions
- Plant Descriptions
- Power Curve
- Soiling Operations
- Tickets
- Tickets from Template
- Variables

6. (Optional) If you want to configure more sheets, repeat steps 1-5 for each sheet.
7. (Optional) If you want the system to automatically send reports to Microsoft SharePoint, click the **Send to SharePoint** checkbox to enable it and fill in these fields:

Send to SharePoint



   a. *SharePoint URL*: enter your organization's Microsoft SharePoint URL.
   b. *SharePoint site*: enter a site within your organization's SharePoint.
   c. *Destination folder*: enter a folder in your SharePoint documents. To create a new folder within the Destination Folder, add a slash and the name of the new folder (example: Reports --> Reports/June).
   d. *SharePoint credentials*: enter your username and password to access SharePoint.

8. (Optional) If you want the system to generate reports automatically, click the **Automatic generation** checkbox and define the automation settings:

a. (Optional) **Delay automatic generation**: If you want to delay the automatic generation, define the amount of hours or days for the delay. This will affect the date and time of the report generation.
b. **Time zone**: select the time zone that will apply to the automatic generation.
c. **Start automatic generation**: select the date at which to begin the automatic generation.

9. Click **Save**.

Result

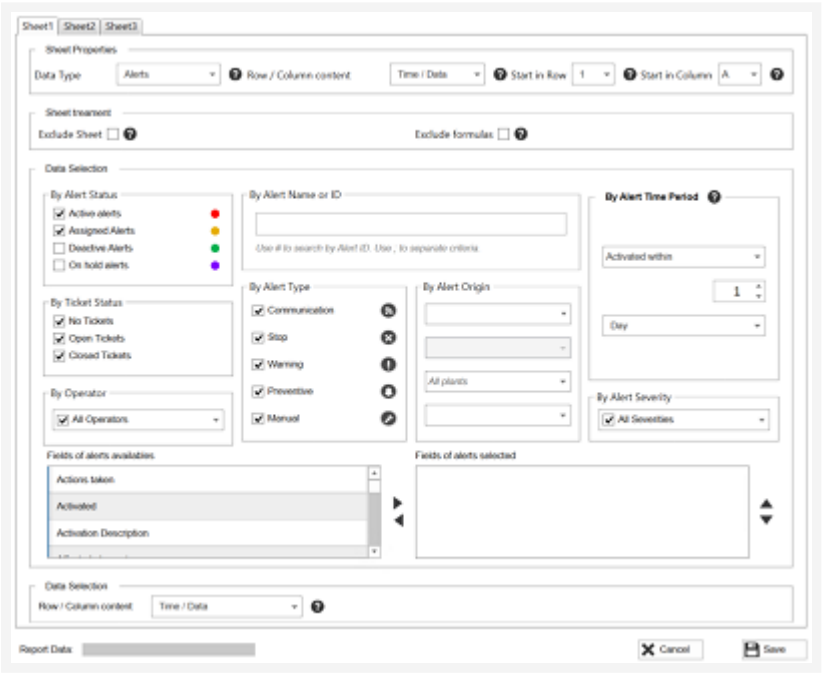
The report is saved and will be generated at the scheduled time.

> ⓘ NOTE: If you want to generate your report immediately, select the report from the Available Templates panel and click **Generate**, then follow the instructions to export the report.

Add alarms to reports

If you selected **Alarms** as a data type, customize the alarm information to include in the report in the **Data Selection** section:

Data Selection (Alarms)



To add alarms to the report, follow these steps:

1. In the **By Alarm Status** section, select the alarm statuses.
2. In the *By Alarm Name or ID* section, enter any larm name or ID. Multiple values must be comma-separated.
3. In the **By Alarm Time Period** section, select the criteria and time period for the alarm.
4. In the **By Ticket Status** section, select the status of the tickets associated to the alarms:
   - **No Tickets**
   - **Open Tickets**
   - **Closed Tickets**
5. In the **By Alarm Type** section, select the alarm types:

- **Communication**
- **Stop**
- **Warning**
- **Preventive**
- **Manual**

6. In the **By Alarm Origin** section, select the plants, devices, or device types to which the alarm is associated.

   ⓘ NOTE: The alarm origin allows you to filter alarms by plants or devices using the pre-defined groups of your portfolio.

7. In the **By Alarm Severity** section, select the alarm severity levels.
8. Select and order the alarm parameters.

   Click the ▶ icon next to the **Available parameters** panel to add them to the **Selected parameters** panel. Click the ◀ icon to remove a parameter from the selection.

   Click the ▲ and ▼ icons to change the order in which the parameters appear in the report.
9. In the **Data Selection** section, select the content to export as rows and the content to be export as columns.
   - **Time/Data:** adds time values as rows and data values as columns.
   - **Time/Data:** adds data values as rows and time values as columns.
10. Click **Save**.

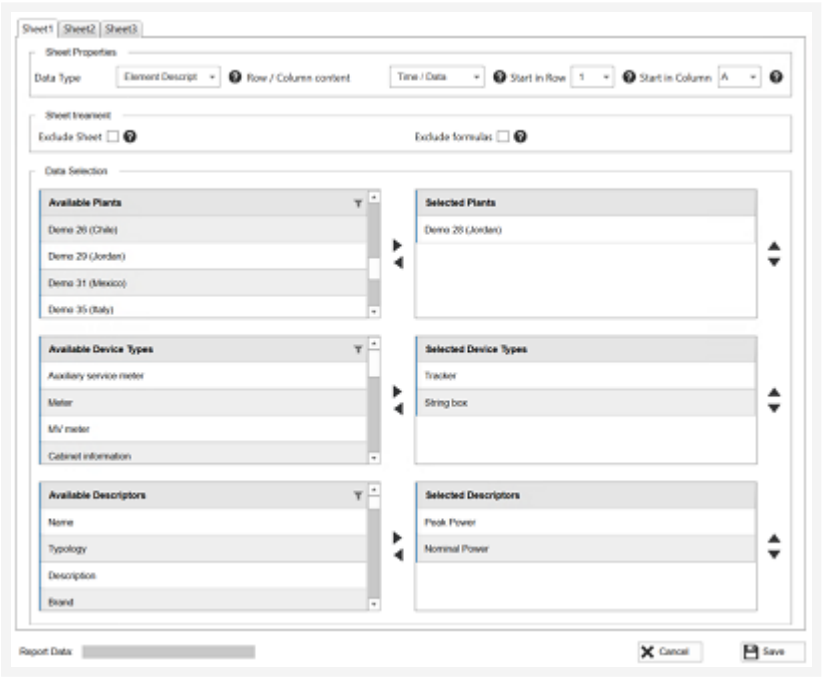
Result

Alarms are added to your report.

Add element descriptions to reports

If you selected **Element descriptions** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Element descriptions)



To add element descriptions to the report, follow these steps:

1. In the **Data Selection** section, select the plants, device types, and descriptors.
   a. Click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Selected Plants** panel.
   b. Click on the device types in the **Available Device Types** panel, then click the ▶ icon to move them to the **Selected Device Types** panel.
   c. Click on the descriptors in the **Available Descriptors** panel, then click the ▶ icon to move them to the **Selected Descriptors** panel.

   Click the ▲ and ▼ icons to change the order in which the selected items appear in the report.
2. Click **Save**.

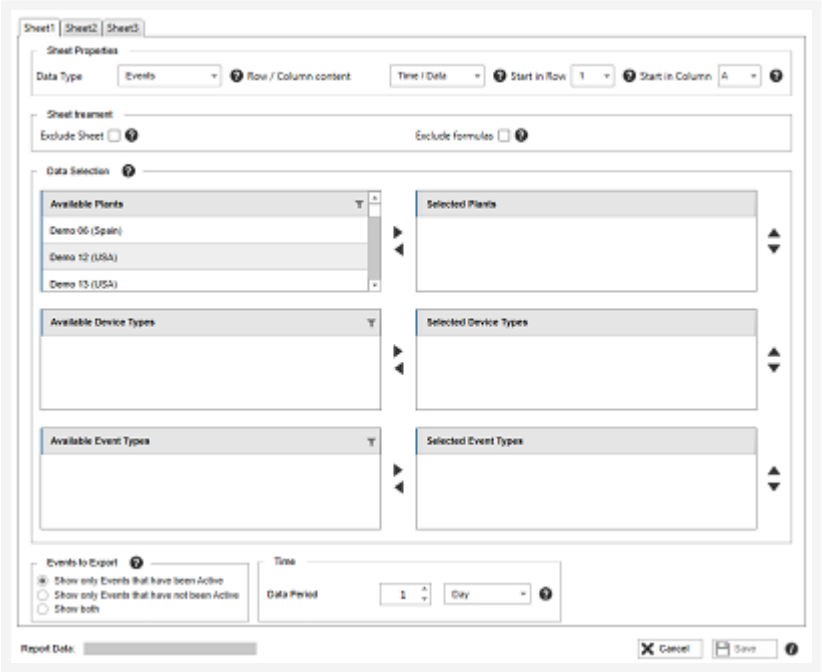
Result

Element descriptions are included in your report.

Add events to reports

If you selected **Events** as a data type, customize the alarm information to include in the report in the **Data Selection** section:

Data Selection (Events)



To add events to a report, follow these steps:

1. In the **Data Selection** section, select the plants, device types, and descriptors.
   a. Click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Selected Plants** panel.
   b. Click on the device types in the **Available Device Types** panel, then click the ▶ icon to move them to the **Selected Device Types** panel.
   c. Click on the descriptors in the **Available Event Types** panel, then click the ▶ icon to move them to the **Selected Event Types** panel.

   Click the ▲ and ▼ icons to change the order in which the selected items appear in the report.
2. In the **Events to Export** section, select the events to include in the report:

- **Show only Events that have been Active**: events whose value changed from 1 to 0.
- **Show only Events that have not been Active**: events whose value changed from 0 to 1.
- **Show both**: all events.

3. In the **Time** section, select the time granularity for the events show in the report.
4. Click **Save**.

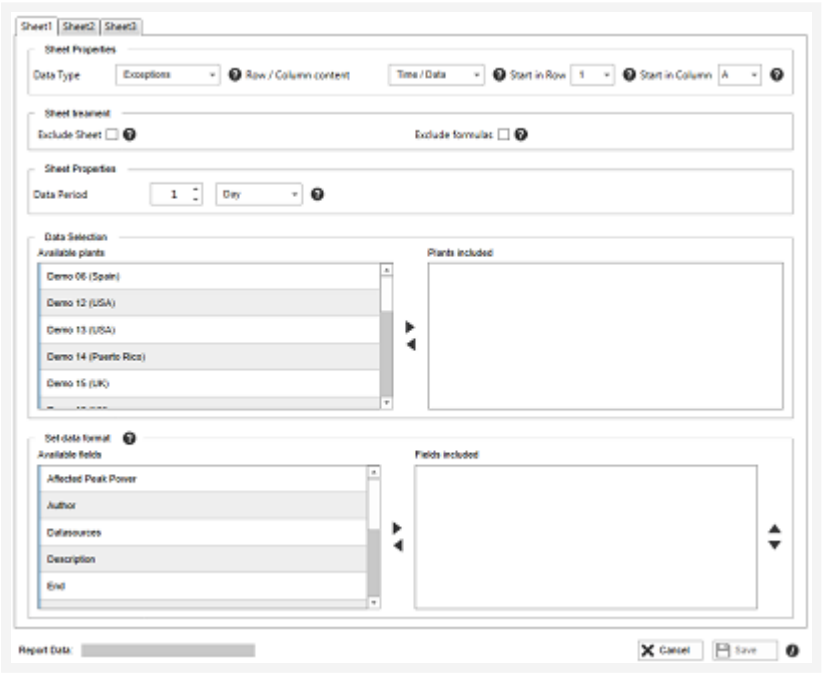
Result

Events are added to your report.

Add exceptions to reports

If you selected **Exceptions** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Exceptions)



To add exceptions to a report, follow these steps:

1. In the **Sheet Properties** section, select the data period.
2. In the **Data Selection** section, select the plants and data format.
   a. Click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Selected Plants** panel.
   b. Click on the device types in the **Set Data Format** panel, then click the ▶ icon to move them to the **Included Fields** panel.

   Click the ▲ and ▼ icons to change the order in which the selected items appear in the report.
3. Click **Save**.

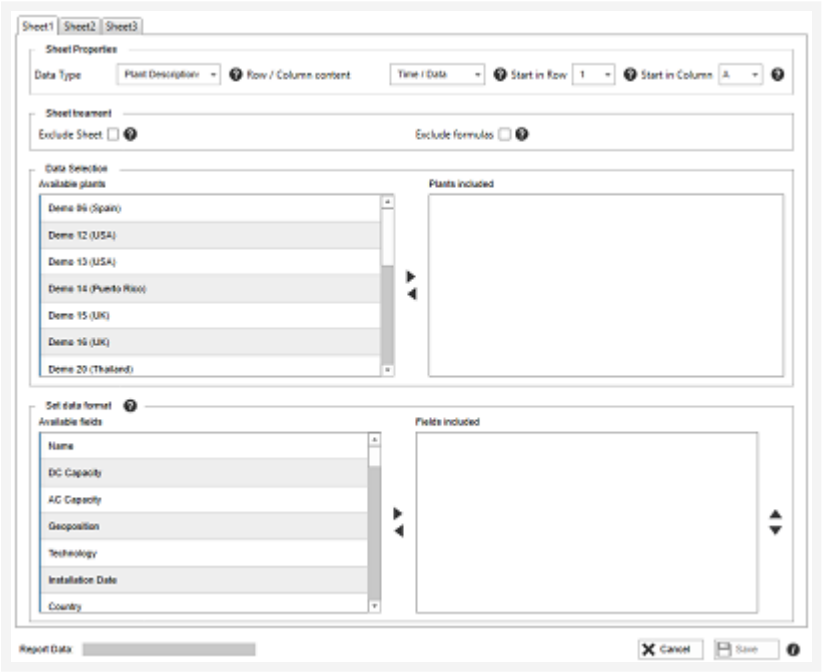
Result

Exceptions are added to your report.

Add plant descriptions to reports

If you selected **Plant descriptions** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Plant descriptions)



To add plant descriptions to a report, follow these steps:

1. In the **Data Selection** section, select the plants, device types, and descriptors.
   a. Click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Selected Plants** panel.
   b. Click on the device types in the **Available Fields** panel, then click the ▶ icon to move them to the **Included fields** panel.

   Click the ▲ and ▼ icons to change the order in which the selected items appear in the report.
2. Click **Save**.

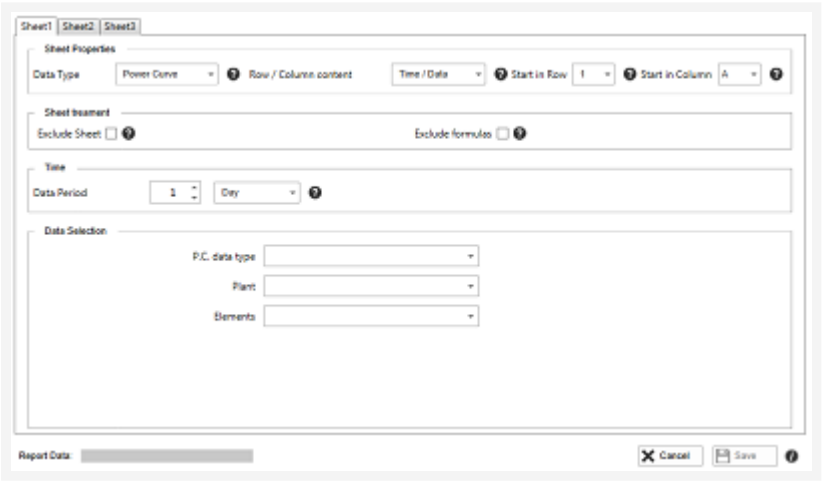
Result

Plant descriptions are added to your report.

Add Power Curve to reports

If you selected **Power Curve** as a data type, customize the alarm information to include in the report in the **Data Selection** section:

Data selection (Power Curve)



To add a power curve to a report, follow these steps:

1. In the **Time** section, select the time period for the power curve.
   - **Fitted and reference data**: the report includes fitted power curve and reference data from the elements you select.
   - **Raw data**: the report includes raw data from the elements you select.
2. In the **Data Selection** section, configure the data for the power curve:
   a. Click the P.C. data type and select an option from the drop-down menu:
   b. Click the **Plant** section and select a Plant from the drop-down menu.
   c. Click the **Elements** section and select an element from the drop-down menu.
3. Click **Save**.

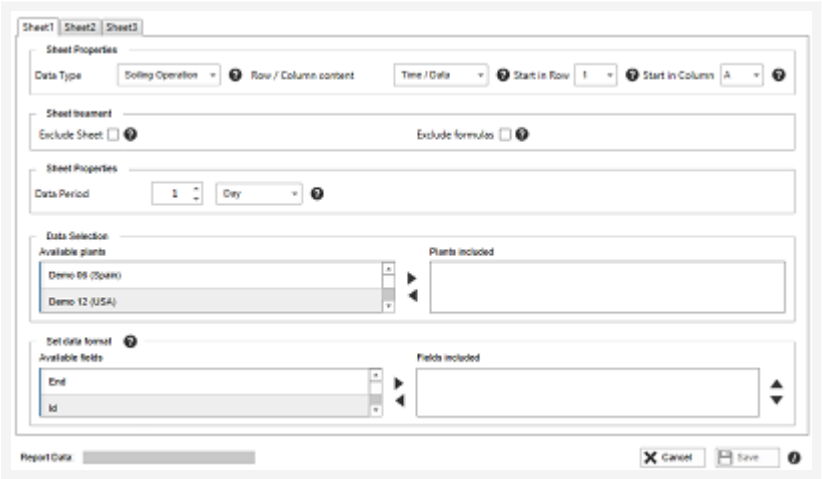
Result

The Power Curve is added to your report.

Add soiling loss operations to reports

If you selected **Soiling operations** as a data type, customize the alarm information to include in the report in the **Data Selection** section:

Data selection (Soiling loss operations)



To add alarms to the report, follow these steps:

1. In the **Data Selection** section, select the data and its format.
   a. Click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Selected Plants** panel.
   b. Click on the device types in the **Set data format** panel, then click the ▶ icon to move them to the **Included fields** panel.

   Click the ▲ and ▼ icons to change the order in which the selected items appear in the report.
2. Click **Save**.

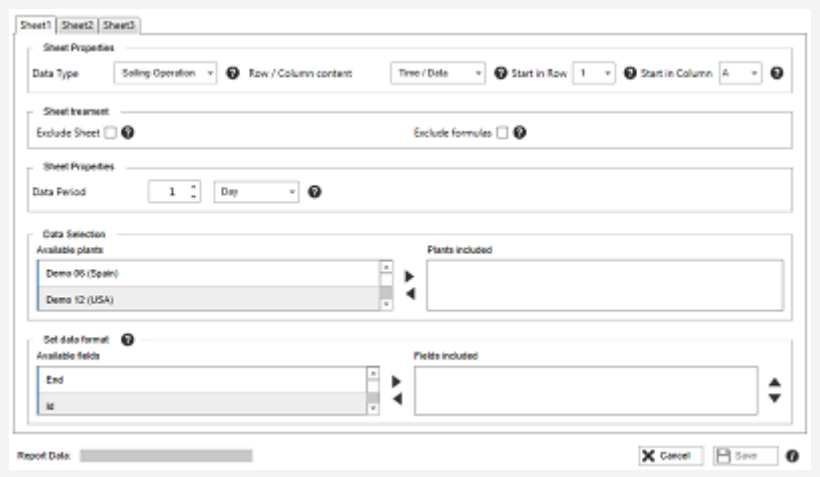
Result

Soiling loss operations are added to your report.

Add tickets to reports

If you selected **Tickets** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Tickets)



> ⓘ NOTE: The report includes only tickets that are scheduled to start within the time period you define for the report.

To add element descriptions to the report, follow these steps:

1. In the **Sheet Properties** section, select the time period for the data.
2. In the **Data Selection** section, click on the plant names in the **Available Plants** panel, then click the ▶ icon to move them to the **Included Plants** panel.
3. Click the Section to Export drop-down menu to select the section of the ticket you want to export.

   > ⓘ BEST PRACTICE: You can export one ticket section per sheet. If you want to export multiple sections, try to export each section on a different sheet.

4. Click **Save**.

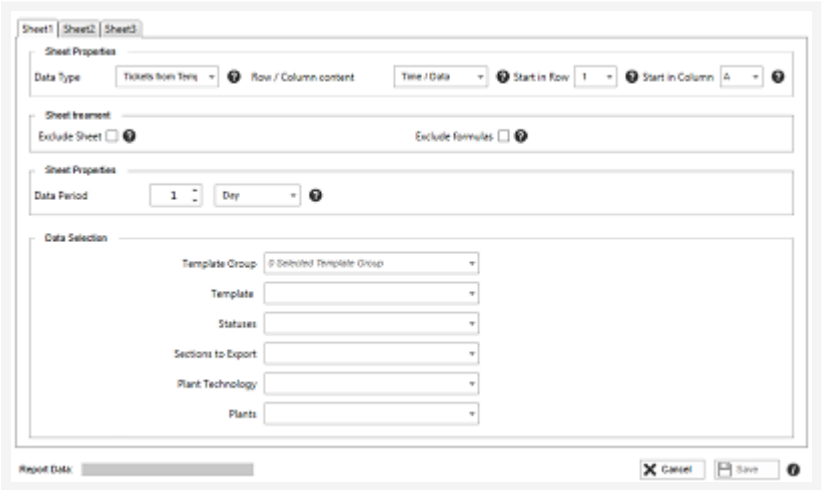
Result

The tickets are added to your report.

Add tickets from templates to reports

If you selected **Tickets from Template** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Tickets from templates)



To add tickets from a template to your report, follow these steps:

1. In the **Sheet Properties** section, select the time period for the data.
2. In the **Data Selection** section, click to select the template and the details of the ticket to include in the report:
   a. **Template**: template from which to retrieve the ticket.
   b. **Statuses**: statuses of the tickets to include in the report.
   c. **Sections to export**: sections from the ticket to include in the report.
   d. **Plant technology**: technology of the tickets to include in the report.
   e. **Plants**: plants to include in the report.
3. Click **Save**.

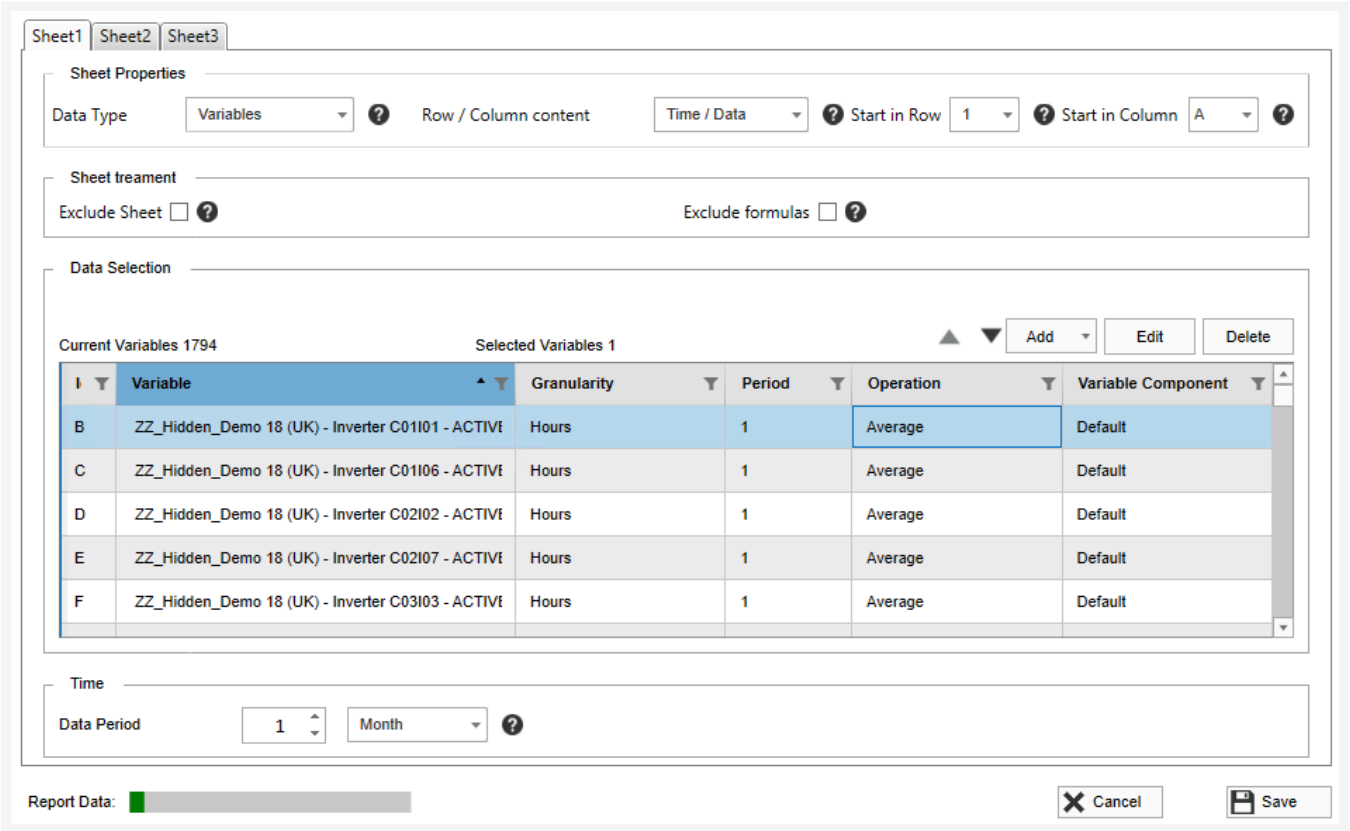
Result

Tickets that match the selected options are added to your report.

Add variables to reports

If you selected **Variables** as a data type, customize the element descriptions to include in the report in the **Data Selection** section:

Data selection (Variables)

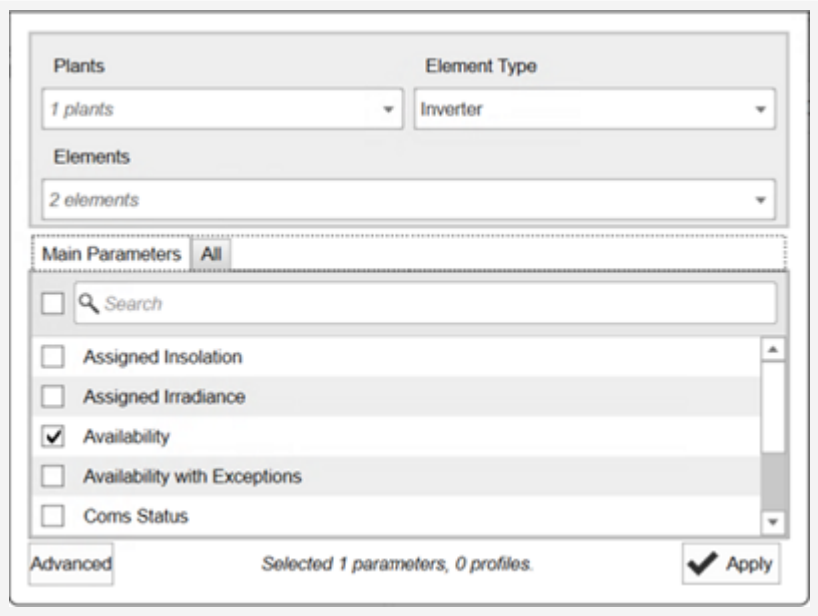


To add variables to a report, follow these steps:

1. In the **Data Selection** section, click the **Add Button** and select where to include the variable in the table.

Result: The Add Variable dialog opens:

Add Variable dialog



2. Select the parameters to add:
   a. Click the **Plants** drop-down menu to select one or more plants. You can use the *Search* field to refine the list.
   b. Click the **Element type** drop-down menu to select one or more element types.

      ⓘ NOTE: You can only select element types after selecting the plants.

   c. Click the **Element** drop-down menu to select one or more elements.

      ⓘ NOTE: You can only select elements after selecting the element types.

   d. Click the **Parameters** tabs to select one or more parameter or series. You can use the *Search* field to refine the list.
   e. (Optional) Click **Advanced Mode** to open the **Advanced Datasource Selector**. For further information, see the Advanced Datasource Selector section.
3. Click **Apply**.
4. In the **Properties** section, customize the variable information by selecting options from the drop-down menu:
   a. Click the Granularity drop-down menu and select the level of granularity for the data (for example, **Hours**).
   b. Click the Period drop-down menu to select the number of time periods to include

in the report (for example, **5** hours).

c. Click the **Data aggregation method** drop-down menu to select the calculation that applies to the data (for example, **Average**).

5. Click **Apply**.
   Result: The Add Variable dialog closes.
6. In the **Data selection** window, click **Save**.

Result

The variables are added to your report.

Soiling loss operations

Soiling is the presence of snow, dirt, soil, dust and other particles on the surfaces of your solar panels. It is one of many factors that can affect the irradiance received by the solar panels in your plant.

"Soiling loss" is the industry term for the monetary losses caused by soiling. Registering soiling loss operations allows GPM SCADA to take into account the real conditions of the panels in your plant. This allows you to have more accurate KPIs.

The system calculates losses by comparing the irradiance values of soiled panels with two panels that serve as references. The first reference panel is one that receives daily cleaning. The second reference panel is a specific panel from each zone of your plant.

You can register soiling loss operations to inform GPM SCADA of when the cleaning of a panel occurs. This also serves to calculate the offset that panels have due to the manufacturing process, aging, or just because they are different models.

Soiling losses are expressed as percentages. The system calculates them using the following formula:

$$\text{SOILING LOSS}(\%) = \left(1 - \frac{Insolation\ Dirty\ Ref.Cell\ *\ Offset}{Insolation\ Clean\ Ref.Cell}\right) * 100$$

For the calculation to work, you must enter three values in the application:

- the cleaning operation for the reference panel.
- the cleaning operation for the reference zone.
- an offset calculation.

You can track and manage all soiling loss operations from the Soiling Loss panel in the Plant

Dashboard module.

Tickets

Tickets allow you to create, assign, and track tickets related to maintenance, operations, and management (O&M) tasks. You can create and manage tickets in the Tickets module. For more information, see the section on Working with tickets.

> ⓘ NOTE: Depending on your setup, one or more customized tickets may be available.

Types of tickets

- Maintenance: track and resolve issues or conditions in your plant, particularly those related to active alarms. When creating maintenance tickets, you can choose between a ticket for a single occurrence, or a series of tickets for recurring tasks. The procedures are the same, except for the first step in which you choose between the two options.
- Task: assign tasks to operators in a plant.
- Data correction: correct data by importing files with information and adding it to the system's Data Provider. This allows you, for example, to create and merge layers of data using Data Sources, to eliminate negative values. You can also create advanced algorithms that calculate final values from sources of raw data.

  > ⓘ NOTE: These tickets require a special configuration. Contact your GPM representative for more information.

Work with tickets

There are two main types of tasks related to tickets: creation and management.

Create tickets

You can create individual maintenance tickets or a series of tickets for recurring tasks.

> ⓘ NOTE: Information contained in Maintenance tickets is customizable. Each customization is stored as a template. The following instructions use the **GPM Default** template.

You can also create tickets from existing tickets. When you create a new ticket from an existing one, you also choose which relational link to create between the two tickets. Additionally, all the general information from the source ticket is copied to the new ticket, saving you time by filling all the mandatory fields.

Create and edit tickets in bulk

It is also possible to create tickets in bulk. This requires exporting the template that corresponds to the tickets and configuring it in Microsoft Excel, and then re-importing it to GPM SCADA. For more information, see Creating and editing tickets in bulk.

Manage tickets

Depending on the type of ticket, there are several other actions you can take:

> ⓘ NOTE: Click on any item in the list to see the instructions to perform that action.

- Add information to tickets:
  - Add interventions to tickets: track physical actions performed on site to fix an issue.
  - Add conversations to tickets: provide additional comments and information regarding the ticket. You can add them as notes.
  - Link tickets: establish relationships between them. This relationship is useful to track tickets when multiple tickets are required to resolve the same issue. For more information, see the table on Ticket relationship types.
  - Edit tickets in bulk: download multiple tickets in an XLS file to easily edit several fields and re-import them into the system.
- Archive tickets: hide tickets from the list when they no longer require your attention. This helps to reduce the loading time of the tickets list.

  > ⓘ NOTE: This action requires an administrator password.

  It is possible to restore archived tickets in order to edit them. You can access archived tickets by toggling the archiving option in the Filtering Options panel.
- Restore archived tickets: enable editing for tickets that were previously archived.

  Restoring is only possible for archived tickets, which you can display by toggling the archiving option in the Filtering Options panel.
- Delete tickets: permanently remove tickets and the information they contain from the application. You can only delete archived tickets.

  > ⓘ NOTE: This action requires an administrator password.

  > ⓘ NOTE: You can delete any additional information section in a ticket by clicking **Delete Section** at the bottom of that section.

Configure tickets

GPM's ticketing system is highly customizable, allowing organizations to adapt them to their different operations and management (O&M) processes.

There are three main areas of work to configure tickets:

- Ticket fields: define the information that appears in the ticket.
- Ticket statuses: define the stages in the workflow of a ticket.
- Ticket templates: define the information (fields) that appear on a particular type of ticket, and the workflow (transitions between statuses). You can also configure tickets templates for use with the GPM Ticket Manager app.

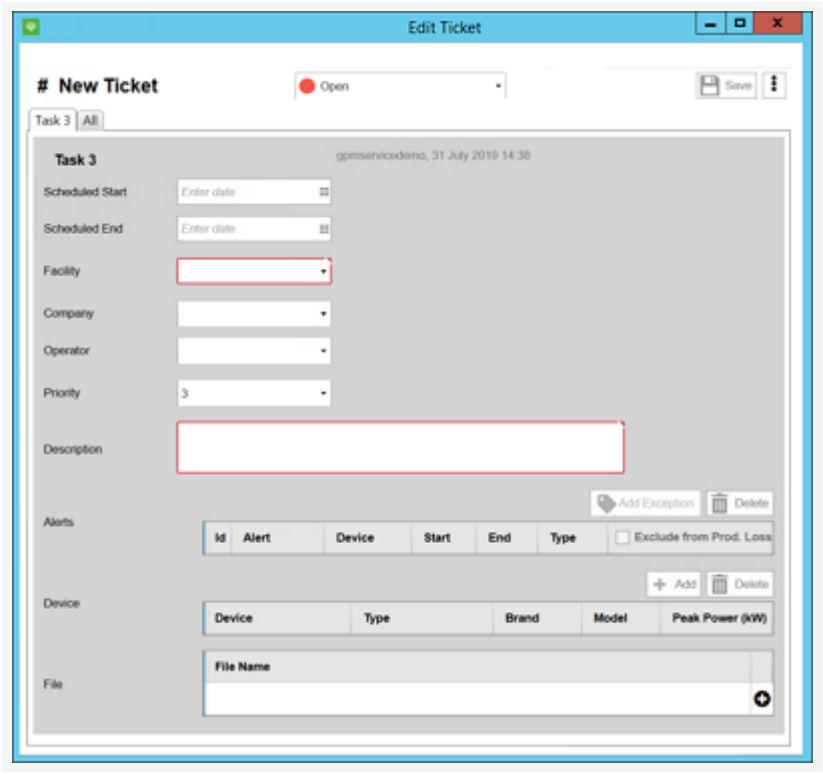
Create maintenance tickets

To create a maintenance ticket, follow these steps:

> ⓘ NOTE: Information contained in Maintenance tickets is customizable. Each customization is stored as a template. The following instructions use the **GPM Default** template.

1. In the Tickets module, click the ⊕ icon and select **Single occurrence**.
2. In the menu, select **Maintenance**, then select the **GPM Default** template.
   Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog



3. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.

e **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f **Priority**: select a priority from the drop-down list.
Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

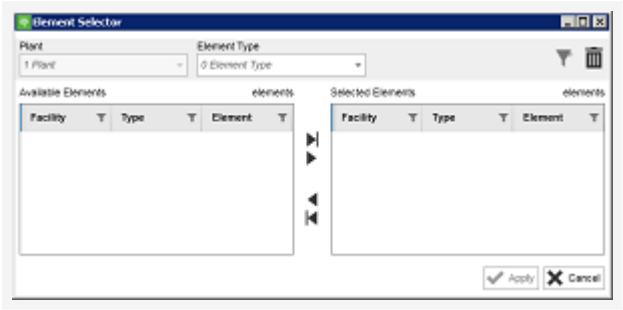
h **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

i (Optional) **File**: click the icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

4 (Optional) In the Device section click **+ Add** to link the ticket to specific elements:
The Element selector dialog appears:

Element selector



a Click the **Element Type** drop-down menu and select the types of element you want to add, then select the element types.

> ⓘ BEST PRACTICE: You can type a term into the *Search* field and click the icon to narrow down the options available on the list.

b In the **Available Elements** panel select the element you want to add to the ticket, then click the icon.
OR: Click the icon to select all the elements on the list.

> ⓘ TIP: Hold down the shift key to select multiple elements.

c Click **Apply**.
Result: The element is added to the list.

5 (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).

6 Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

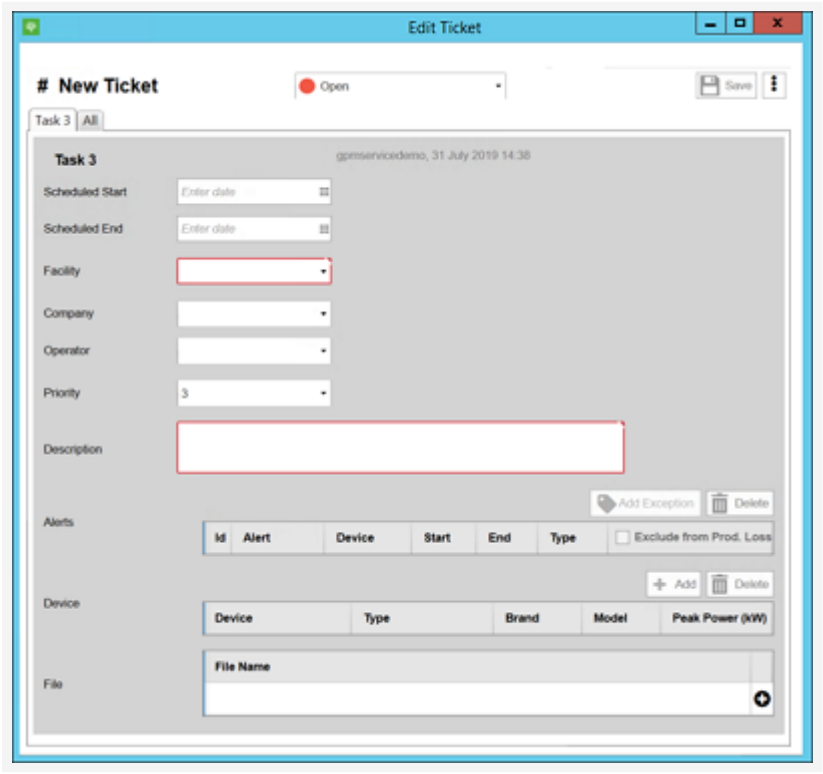
Create series of maintenance tickets

To create a series of maintenance tickets, follow these steps:

> ⓘ NOTE: Information contained in Maintenance tickets is customizable. Each customization is stored as a template. The following instructions use the **GPM Default** template.

1. In the Tickets module, click the ⊕ icon and select **Series**.
2. In the menu, select **Maintenance**, then select the **GPM Default** template.
   Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog



3. In the **Edit Ticket** dialog, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.

e **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f **Priority**: select a priority from the drop-down list.
Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

h **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

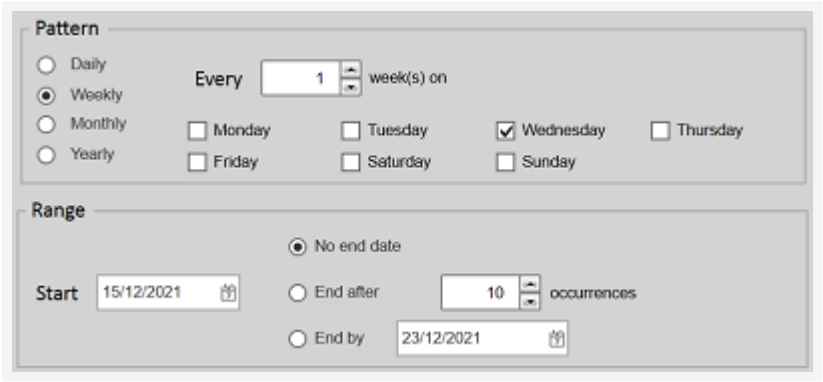
i (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

4 (Optional) Change the status of the ticket from the drop-down list (for example, **Open**).

5 Click the **Recurrence** tab.

Recurrence tab



6 In the **Pattern** section of the Recurrence tab, configure the frequency of the recurrence:

a Select a period for the recurrence (for example, **Weekly**).

b Select a frequency for the recurrence (for example, "Every **1** week").

c Select the day of the week on which the occurrence takes place (for example, **Wednesday**).

7 In the **Range** section of the Recurrence tab, select the dates on which the recurrence starts and ends.

8 Click **Save**.

Result

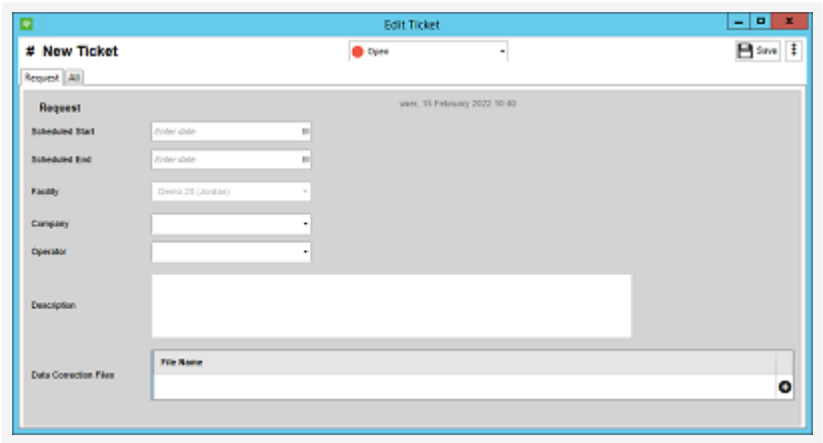
The series of tickets is created and assigned to the operator you selected.

Create data correction tickets

To create a data correction ticket, follow these steps:

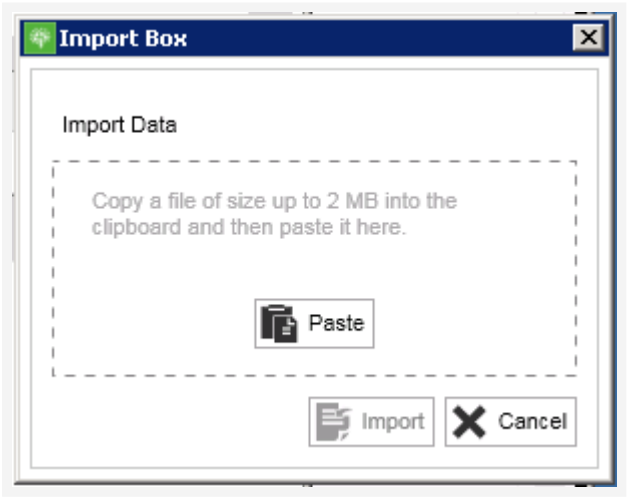
1. In the Tickets module, click the ⊕ icon and select **Single occurrence**.
2. In the menu, select **Data Cleansing**, then select **GPM Data Correction**.
   Result: The Edit ticket dialog appears:

   ### Edit ticket dialog (Data cleansing)

3. In the Request section, enter the information for the ticket:
   a. **Scheduled Start**: select the date on which the ticket starts.
   b. **Scheduled End**: select the date on which you expect the ticket to close.
   c. **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.
   d. **Company**: select the company that must resolve the issue related to the ticket.
   e. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   f. *Description*: enter a description in the text input field.
4. In the Data Correction Files section, click the ⊕ icon to open the Import Box and import an XLS file from which to add the corrected data:

Import Box



ⓘ NOTE: You can only import XLS files.

a. In your computer's File Explorer, select and copy the file you want to import by pressing CTRL + C, or by right-clicking the file and selecting **Copy**.

b. In the Import Box dialog, click **Paste**.

c. Click **Import**.

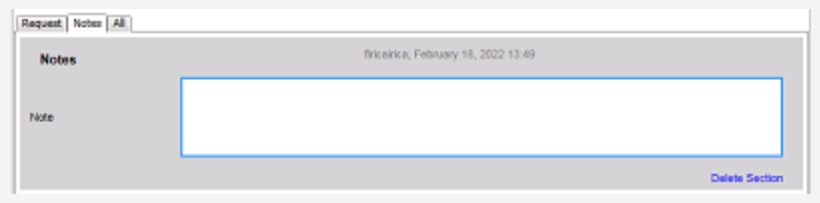
Result: The file is imported to the ticket.

5. (Optional) To add a note, follow these steps:

a. Click the ⋮ icon, hover over **Add section** and select **Note**.

Result: The Note tab appears.

Note tab in ticket edition



b. Enter text in the *Note* field and click **Save**.

Result: The note is added to the ticket.

6. Click **Save**.

Result

The ticket is created and assigned to the operator you selected.

Create new tickets from existing tickets

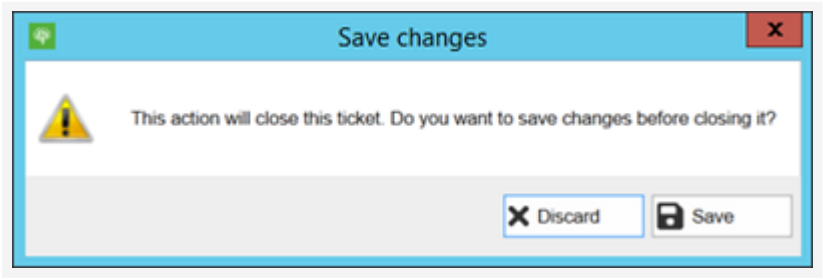
To create a new ticket from an existing ticket, follow these steps:

1. Open the ticket that you want to use as the source ticket.
2. Click the ⋮ icon and go to **Add New Ticket**, then select the relationship and the ticket type.

   > ⓘ NOTE: Alternatively, you can create a new ticket from an existing ticket from the context menu in the ticket list.

   Result: The **Save changes** dialog appears:

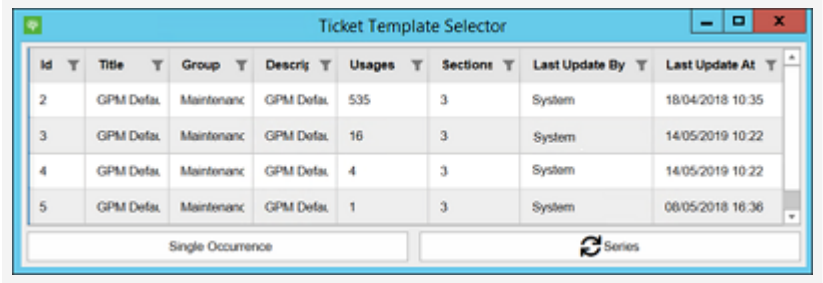
   ### Save changes dialog

3. On the **Save changes** dialog, select whether you want to save your latest changes to the source ticket.

   Result: The **Ticket Template Selector** appears:

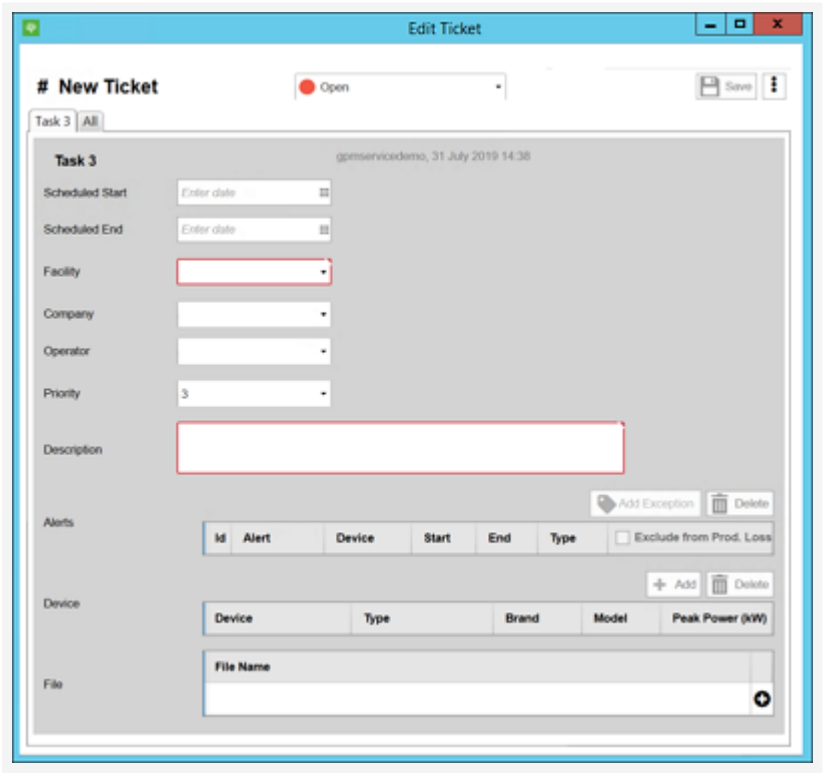
   ### Ticket Template Selector

4. On the **Ticket Template Selector** dialog, select the ticket template, and click **Single Occurrence**.

Result: The **Edit Ticket** dialog appears:

Edit Ticket dialog



5 In the **Edit Ticket** dialog, enter the information for the ticket:

a **Scheduled Start**: select the date on which the ticket starts.

b **Scheduled End**: select the date on which you expect the ticket to close.

c **Facility**: if your portfolio has more than one plant, select a plant from the drop-down list.

d **Company**: select the company that must resolve the issue related to the ticket.

e **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.

f **Priority**: select a priority from the drop-down list.

Priority is calculated on a scale from one to five, where one is the highest priority and five is the lowest priority.

g *Description*: enter a description in the text input field.

h **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

i (Optional) **File**: click the ⊕ icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when creating a maintenance ticket.

6 When you have entered all the details, click 💾 **Save.**

Result

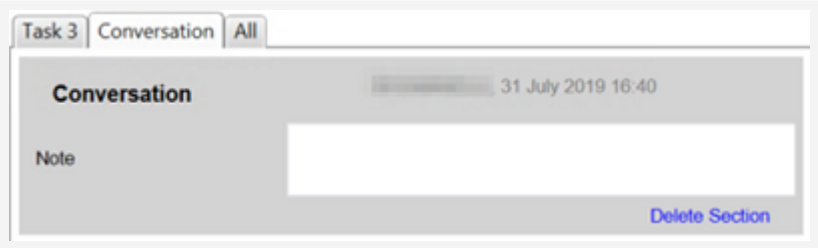
A new ticket is created and a link is generated between the two tickets.

Add conversations to tickets

To add a conversation to a ticket, follow these steps:

1. Open the ticket to which you want to add the conversation.
2. Click the ⋮ icon and select 💬 **Add Conversation**.
   Result: The **Conversation** tab appears:

   ### Conversation dialog

3. In the *Note* section of the **Conversation** tab, enter the text you want to add.
4. Click 💾**Save**.

Result

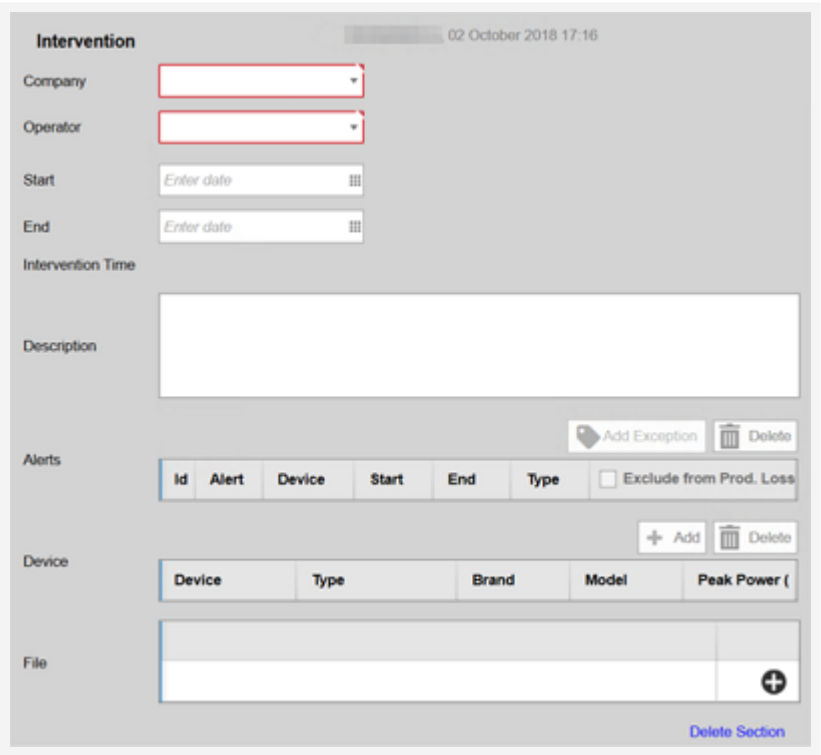
The conversation is added to the ticket.

Add interventions to tickets

To add an intervention to a ticket, follow these steps:

1. Open the ticket to which you want to add the intervention.
2. Click the ⋮ icon and select **Add Intervention**.
   Result: The **Intervention** dialog appears:

Intervention dialog



3. In the **Intervention** dialog, enter the information for the intervention:
   a. **Company**: select the company that must resolve the issue related to the ticket.
   b. **Operator**: select the username of an operator from the drop-down list to assign the ticket to them.
   c. **Start**: select the date on which the intervention starts.
   d. **End**: select the date on which you expect the intervention to end.
   e. *Description*: enter a description for the intervention in the text input field.
   f. **Device**: if you have selected a plant, you can specify the elements to which the ticket applies

g **File**: click the icon to add files to the ticket. For more information, see Import data from a file.

> ⓘ NOTE: The **Alarms** field is unavailable when adding interventions to a maintenance ticket.

4 Click **Save**.

Result

The intervention is added to the ticket and assigned to the operator you selected.

Link tickets

To link a ticket to another ticket, follow these steps:

1. Open the ticket you want to link.
2. Click the ⋮ icon and go to **Link to Ticket**.

   ⓘ NOTE: Alternatively, you can add a link to a ticket from the context menu in the ticket list.

3. In the **Link to Ticket** menu, select the relationship type you want to establish with the link:

Relationship type	Description
Hierarchical	Tree structure between tickets, where a parent ticket contains one or more child tickets: ▪ Parent: ticket contains other entities. ▪ Child: ticket has a parent entity.
Sequential	Temporal order in which tickets should be completed: ▪ Predecessor: ticket should be completed before another ticket. ▪ Successor: the ticket should be completed after another ticket.
Related	Non-directional relationship between tickets.

4. In the *Ticket ID* field, enter a valid ticket ID and press **Enter**.
5. Click **Save**.

Result

A link is created between the two tickets. The link appears in the **Links** tab of the two tickets and in the **Links** column of the **Tickets** module.

Archive tickets

Before you begin

This task requires an administrator password.

To archive a ticket, follow these steps:

1. On the ticket list, select the tickets you want to archive.
2. Right-click the selection and select **Archive** from the context menu.
   Result: The **Security Validation** dialog appears.
3. On the **Security Validation** dialog, enter the administrator password and click **Accept**.

Result

The tickets are archived and hidden from the ticket list.

Restore archived tickets

To restore archived tickets, follow these steps:

1. In the Tickets module, expand the Filtering options panel and set the toggle to **Archived**.
   Result: The tickets list appears.
2. On the tickets list, select the tickets you want to restore and right-click them to open the context menu.
3. In the context menu, click **Restore**.

Result

The tickets are restored and become available in the default ticket list.

Delete tickets

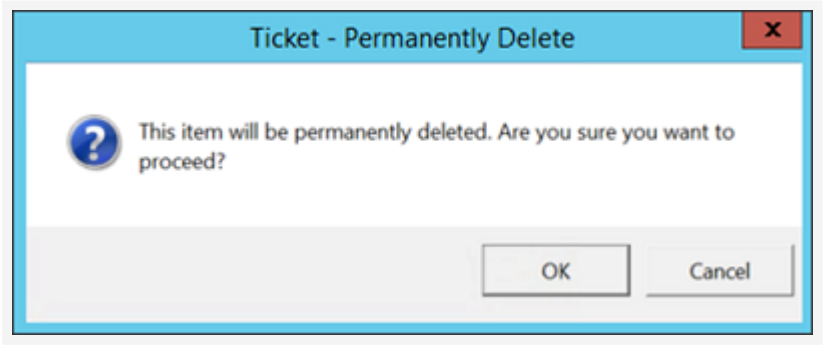
Before you begin

This task requires an administrator password.

To delete tickets, follow these steps:

1. In the Tickets module, expand the filtering options panel and set the toggle to **Archived**.
2. On the ticket list, select the tickets you want to delete.
3. Right-click the selection, then select **Delete** from the context menu.
   Result: The **Confirmation** dialog appears:

   ### Confirmation dialog

4. In the **Confirmation** dialog, click **OK**.
   Result: The **Security Validation** dialog appears.
5. In the **Security Validation** dialog, enter the administrator password and click **Accept**.

Result

The tickets are deleted and the information they contain is permanently removed from the application.

Creating and editing tickets in bulk

To create or edit tickets in bulk, you must export the template that corresponds to the tickets and configure it in Microsoft Excel, and then re-import the XLS file to GPM SCADA.

Create

To create tickets in bulk, you must export a ticket template as an XLS file, which you then edit in Microsoft Excel. When you import the XLS file to GPM SCADA, the tickets are automatically added to the system and assigned to the operators you defined in the configuration.

There are two main methods to create tickets in bulk:

- Create single tickets in bulk
- Create a series of recurring tickets in bulk

Edit

To edit tickets in bulk, you first export them as an XLS file, which you then edit in Microsoft Excel. Each ticket has its own unique ID, which together with the reference from the template, allows the system to map the tickets when you import the XLS file to GPM SCADA.

> TIP: You can create new tickets by adding them to the table in the XLS file, and filling in the information for the fields.

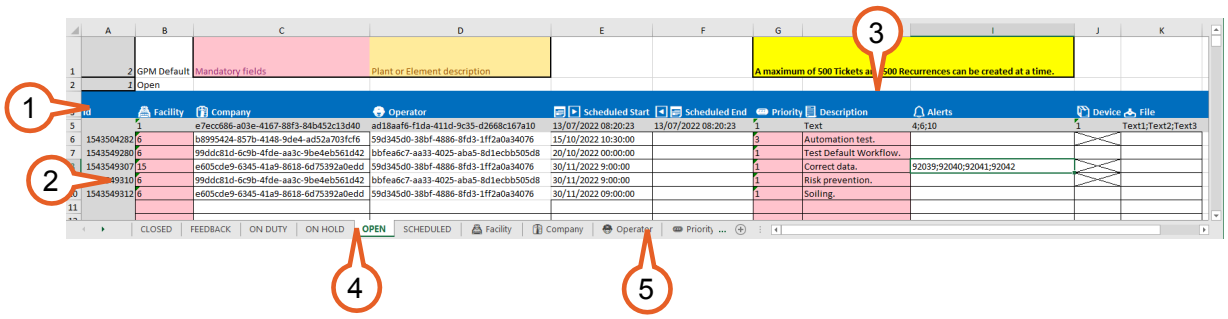
Ticket templates

In Microsoft Excel, column headers display the fields of the ticket template. The row immediately below the column headers provides clear examples of how to fill in each field. This makes it easy to add and edit information, and it allows you to easily add new tickets when editing a list of existing tickets.

Fields

- Mandatory: cells marked in red (for example, "Plant"). Gray rows are blocked or non-editable cells.
- Descriptions: cells marked in yellow. These fields include pre-configured metadata for Plants and Elements, that the system automatically fills in when the XLS file is re-imported to GPM SCADA.
- Pre-defined values: cells with a limited list of available options that correspond to entities in GPM SCADA (for example, "Company" and "Operator"). The XLS file includes references to correctly input these values, which the system automatically maps to the entity when re-importing the file. For more information, see Field reference tabs below.

Ticket template in XLS



1. Ticket fields: each column corresponds to a field in the ticket configuration. The first row provides an example of the format for inputting data.

   ⓘ NOTE: The cells that inform the fields of the tickets must have the same format as the one provided in the template (for example, the **Scheduled Start** column must have date format).

2. Ticket IDs: each ticket has a unique ID automatically assigned to it, to correctly identify it when re-importing the file. When creating new tickets, this field must remain blank.
3. Maximum tickets: the maximum number of tickets you can edit and import at a

time.

ⓘ NOTE: The maximum number of tickets that you can upload at a time depends on your product configuration. For more information, contact your GPM representative.

4. Ticket status tabs: tickets are sorted by status. Click on a tab to see the tickets that have that status (for example, "Open").
5. Field reference tabs: click to see the available values for each field of the template.

Create tickets in bulk

Before you begin

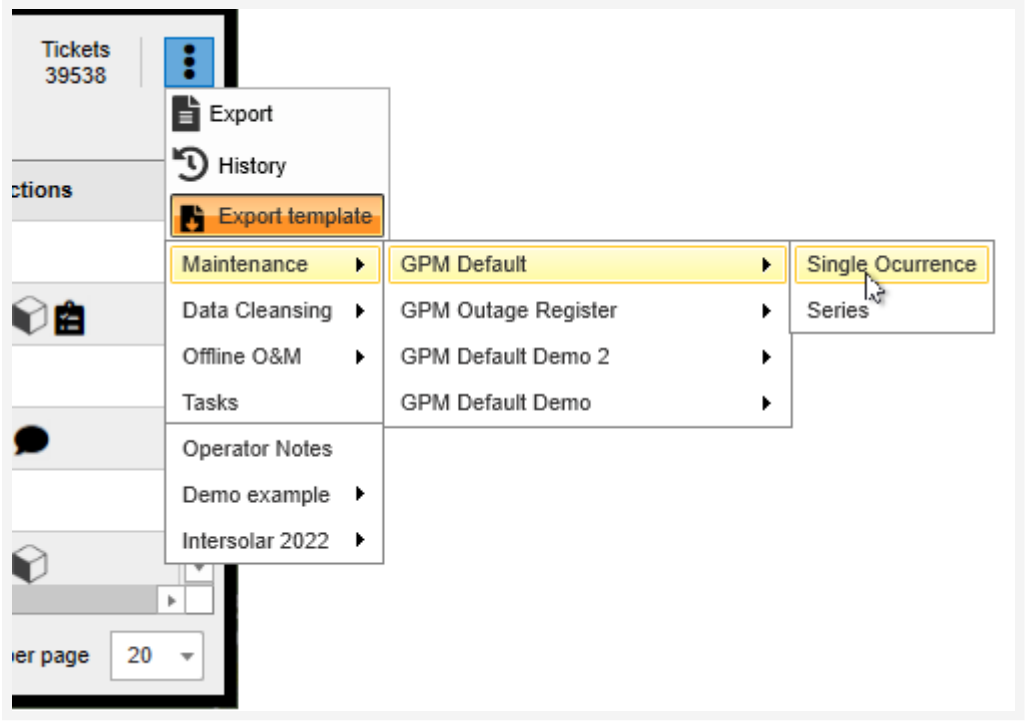
This task requires an administrator password.

To create tickets in bulk, follow these steps:

1. In the Tickets module, click the ⋮ icon to open the actions menu and select **Export template**.

   Result: The Template types menu appears:

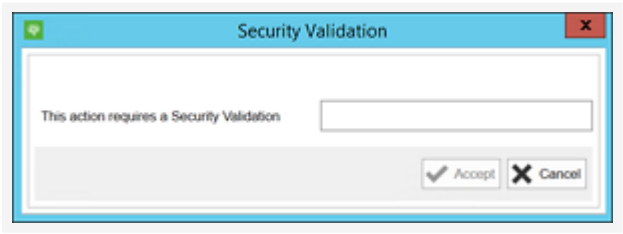
   ### Template types menu

2. Hover over the type of ticket you want to create (for example, **Maintenance**) and select the template you want to export. Then, select **Single occurrence**.

Result: The Security validation dialog appears:

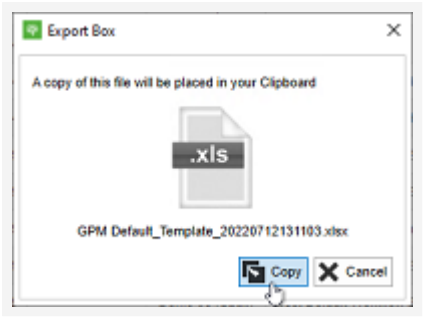
Security validation dialog



3. Enter the administrator password.

   Result: The **Export box** appears:

Export box



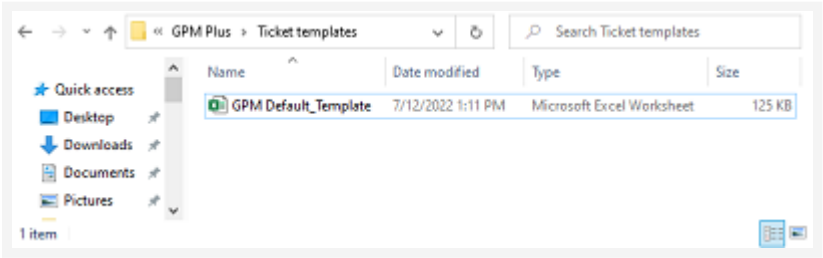
4. Click the **Copy** button.

   Result: The XLS file is copied to your clipboard.

5. In your computer's File Explorer, navigate to the folder in which you want to save the XLS file and press CTRL+V, or right-click and select **Paste**.

   Result: The XLS file is copied to the folder you selected:

File Explorer

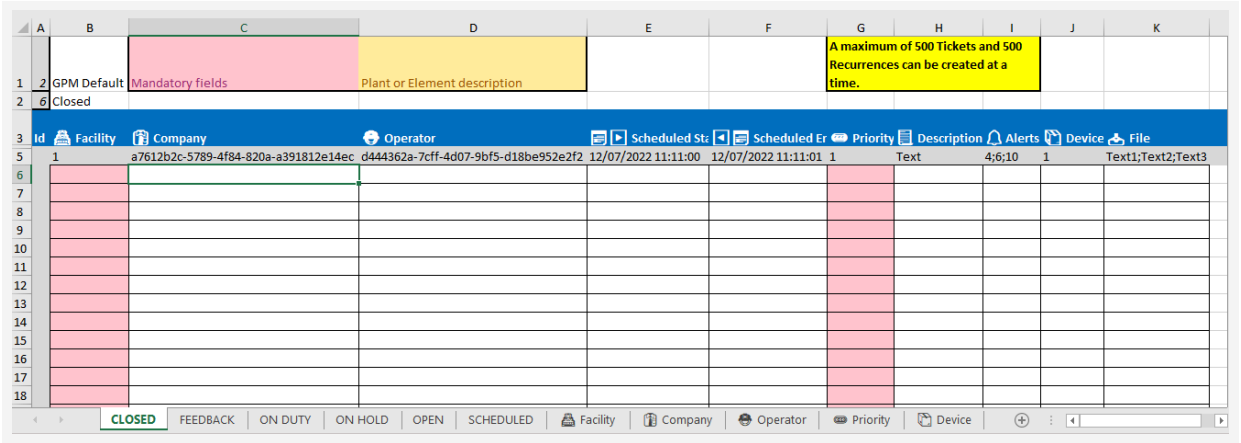


6. Open the XLS file, where each row corresponds to a ticket, and each column corresponds to a field.

ⓘ NOTE: For more information, or to see the fields in context, see the Edit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticket.

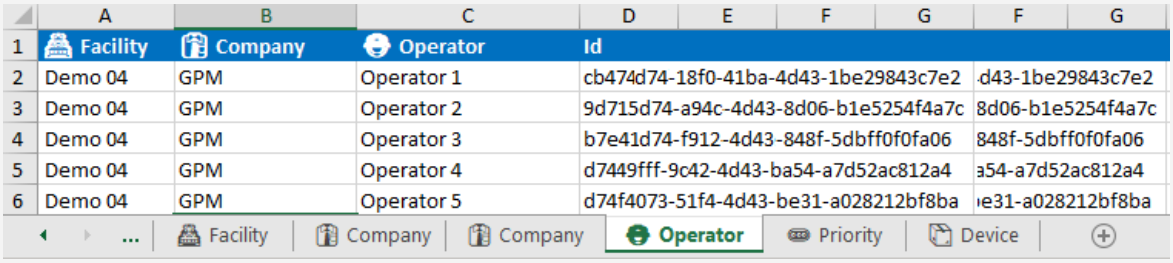
Result:

Ticket template in Microsoft Excel



☆ IMPORTANT: The column headers display examples of the format for each field. Each page in the XLS file provides and ID and a reference for the available options in your system.

Sample IDs and references for the "Operator" field



7 Fill in the columns of the template to configure the corresponding fields.

> ⓘ NOTE: The columns of mandatory fields are marked in red (for example, "Facility" in column 1).

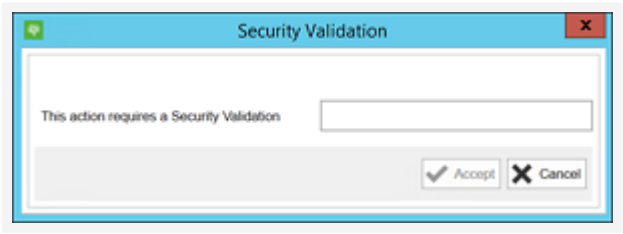
> ⓘ TIP: For each field, you can go to the corresponding page in the XLS file, select and copy the relevant ID, and paste it in the column of the corresponding field.

a. *Facility*: if your portfolio has more than one plant, enter the ID of the plant for which you want to create the ticket.
b. (Optional) *Company*: enter the ID of the company that must resolve the issue related to the ticket.
c. (Optional) *Operator*: enter the ID of an operator to assign the ticket to them.
d. (Optional) *Scheduled Start*: enter the date and time on which the ticket starts, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).
e. (Optional) *Scheduled End*: enter the date and time on which the ticket ends, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).
f. *Priority*: enter a level of priority (for example, 1).
g. *Description*: enter a description for the ticket.
h. (Optional) *Alarms*: enter the ID for the alarms to which you want to link the ticket.
i. (Optional) *Element*: enter the ID for the elements to which you want to link the ticket.
j. (Optional) *File*: enter the IDs of the files to which you want to link the ticket.
k. Repeat sub-steps (a) to (j) for each ticket you want to create.

8. Press **CTRL+S**, or click on **File** and select **Save**.
   Result: The changes to the XLS file are saved.
9. In the Tickets module of GPM SCADA, click the ⊕ icon and select **Create/Edit in bulk**.

Result: The **Security validation** dialog appears:

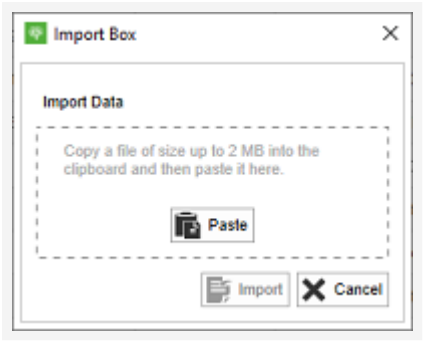
Security validation dialog



10 Enter the administrator password.

Result: The **Import box** appears:

Import box



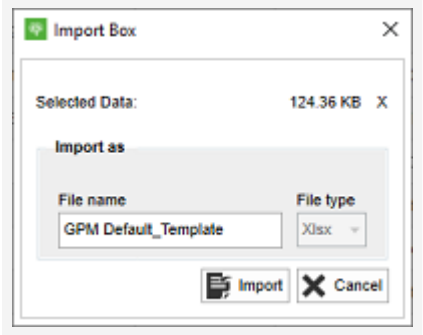
11 In your computer's File Explorer, select the XLS file with the ticket template and press **CTRL+S**, or right-click the XLS file and select **Copy**.

Result: The XLS file is copied to your clipboard.

12 In the Import box, click **Paste**.

Result: The XLS file is copied to the Import box.

Import box

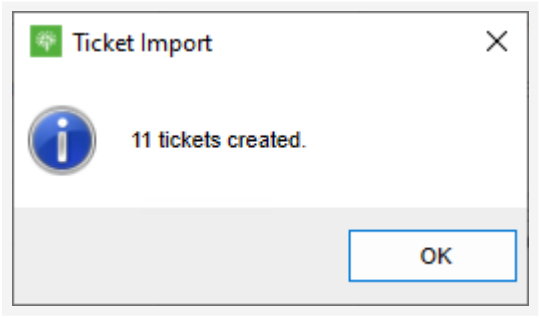


13 (Optional) Enter a new name for the file in the *File name* field.

14 Click **Import**.

Result: The **Ticket Import** dialog appears:

Ticket Import dialog

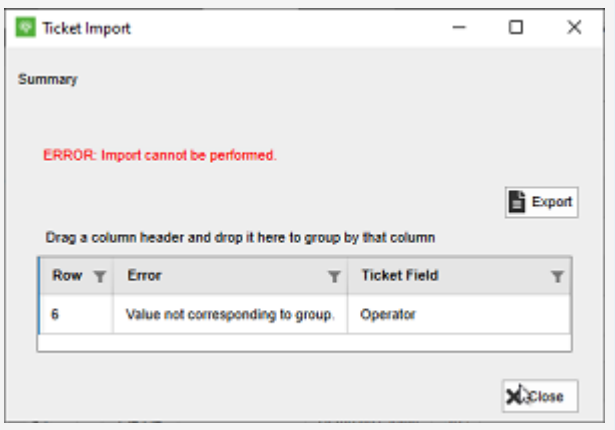


Result

The XLS file is imported and the tickets are added to the system.

ⓘ NOTE: If the import fails, an error dialog appears providing details about the issue. To fix it, go back to Step 7 (above) and correct the errors in the specified fields.

Error message in Ticket Import dialog



Create series of tickets in bulk

Before you begin

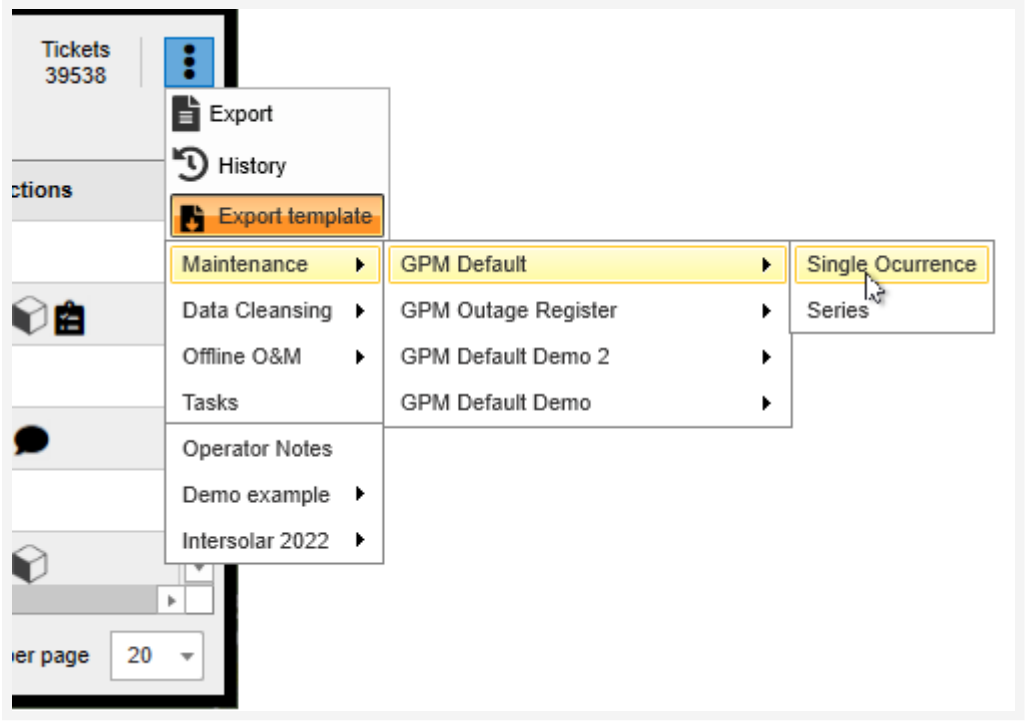
This process requires an administrator password.

To create a series of recurring tickets in bulk, follow these steps:

1. In the Tickets module, click the ⋮ icon to open the actions menu and select **Export template**.

   Result: The Template types menu appears:

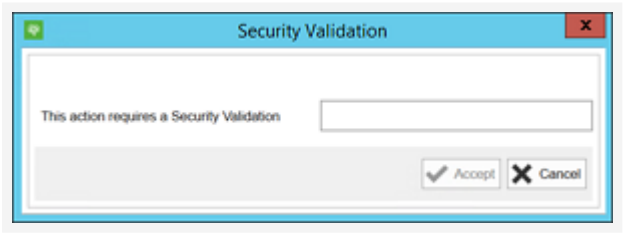
   ### Template types menu

2. Hover over the type of ticket you want to create (for example, **Maintenance**) and select the template you want to export. Then, select **Series**.

Result: The Security validation dialog appears:

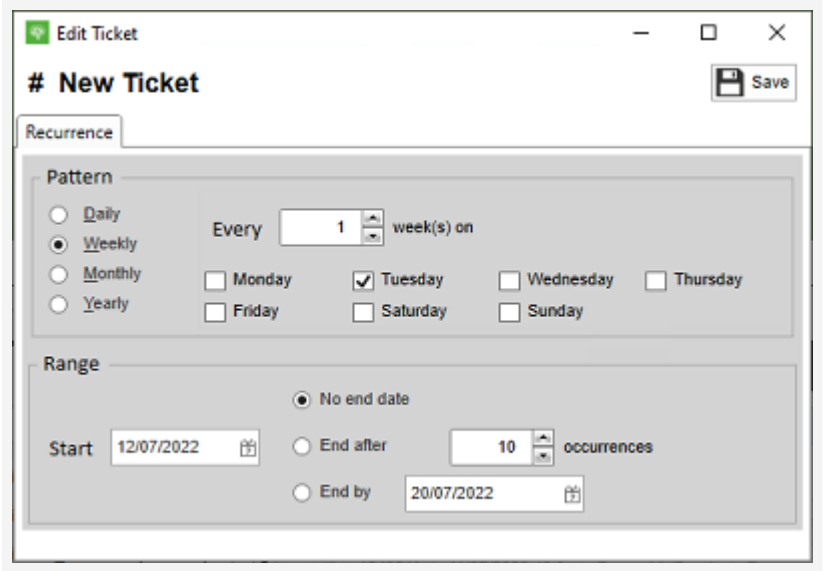
Security validation dialog



3. Enter the administrator password.

Result: The **Recurrence configuration** dialog appears:

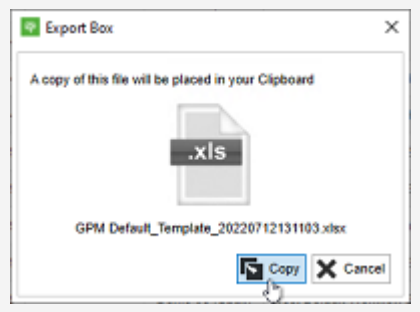
Recurrence configuration dialog



4. In the Recurrence dialog, define how often and for how long the tickets will recur:
   a. In the Pattern section, define the pattern for the recurrence.
   b. In the Range section, define the time period during which the tickets will recur.
   c. Click **Save**.

Result: The export box appears:

Export box



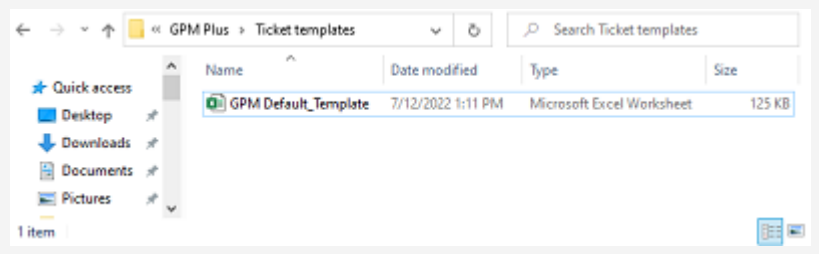
5. Click the **Copy** button.

   Result: The XLS file is copied to your clipboard.

6. In your computer's File Explorer, navigate to the folder in which you want to save the XLS file and press CTRL+V, or right-click and select **Paste**.

   Result: The XLS file is copied to the folder you selected:

File Explorer

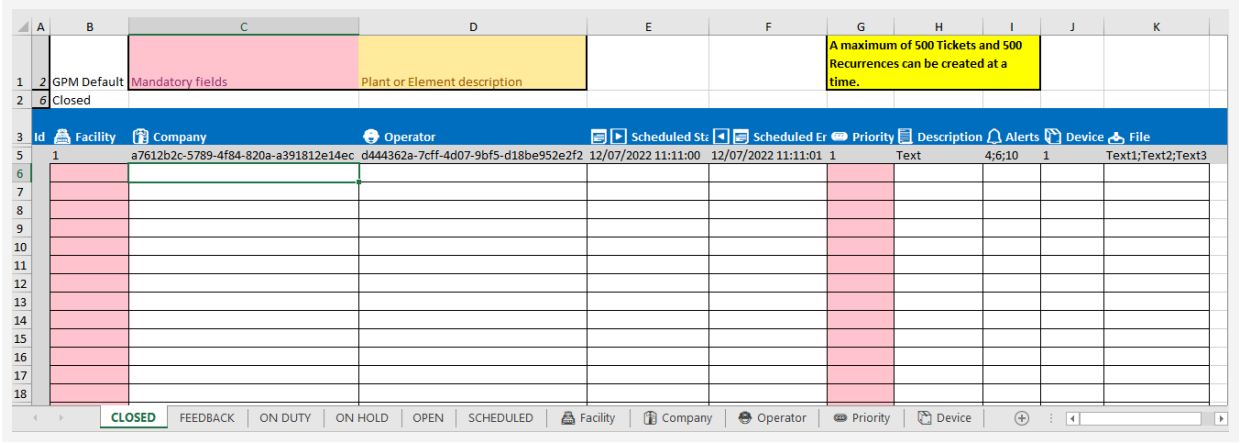


7. Open the XLS file, where each row corresponds to a ticket, and each column corresponds to a field.

> ⓘ NOTE: For more information, or to see the fields in context, see the Edit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticket.

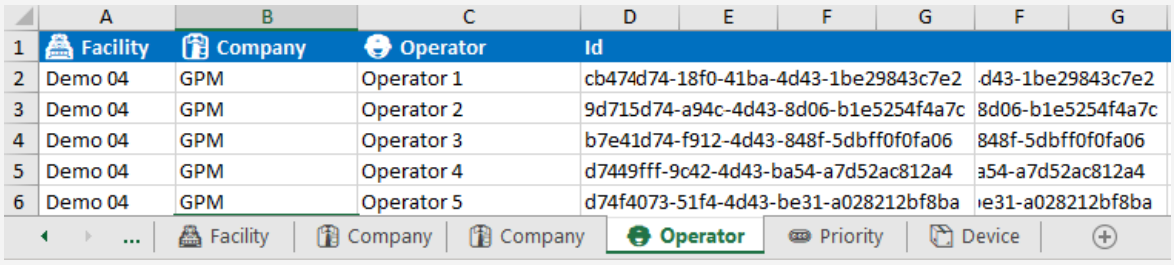
Result:

Ticket template in Microsoft Excel



☆ IMPORTANT: The column headers display examples of the format for each field. Each page in the XLS file provides and ID and a reference for the available options in your system.

Sample IDs and references for the "Operator" field



8 Fill in the columns of the template to configure the corresponding fields.

ⓘ NOTE: The columns of mandatory fields are marked in red (for example, "Facility" in column 1).

⍰ TIP: For each field, you can go to the corresponding page in the XLS file, select and copy the relevant ID, and paste it in the column of the corresponding field.

a *Facility*: if your portfolio has more than one plant, enter the ID of the plant for which you want to create the ticket.

b (Optional) *Company*: enter the ID of the company that must resolve the issue related to the ticket.

c. (Optional) *Operator*: enter the ID of an operator to assign the ticket to them.

d. (Optional) *Scheduled Start*: enter the date and time on which the ticket starts, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).

e. (Optional) *Scheduled End*: enter the date and time on which the ticket ends, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).

f. *Priority*: enter a level of priority (for example, 1).

g. *Description*: enter a description for the ticket.

h. (Optional) *Alarms*: enter the ID for the alarms to which you want to link the ticket.

i. (Optional) *Element*: enter the ID for the elements to which you want to link the ticket.

j. (Optional) *File*: enter the IDs of the files to which you want to link the ticket.

k. Repeat sub-steps (a) to (j) for each ticket you want to create.

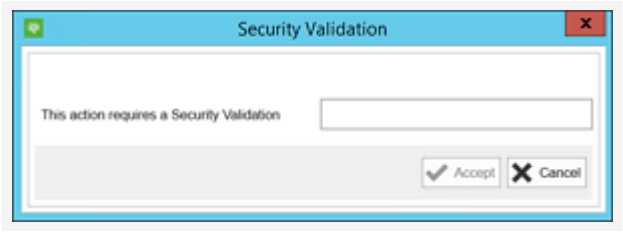
9. Press **CTRL+S**, or click on **File** and select **Save**.

Result: The changes to the XLS file are saved.

10. In the Tickets module of GPM SCADA, click the ⊕ icon and select **Create/Edit in bulk**.

Result: The **Security validation** dialog appears:

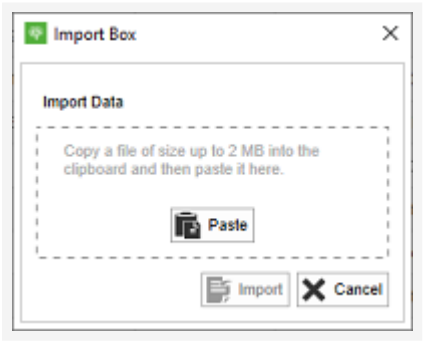
Security validation dialog



11. Enter the administrator password.

Result: The **Import box** appears:

Import box



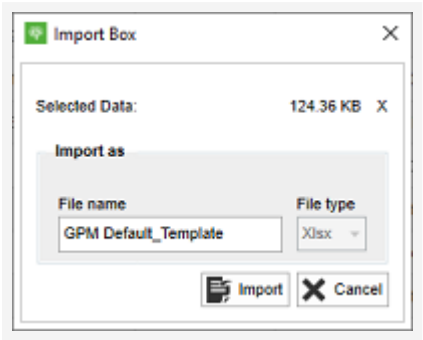
12 In your computer's File Explorer, select the XLS file with the ticket template and press **CTRL+S**, or right-click the XLS file and select **Copy**.

Result: The XLS file is copied to your clipboard.

13 In the Import box, click **Paste**.

Result: The XLS file is copied to the Import box.

Import box

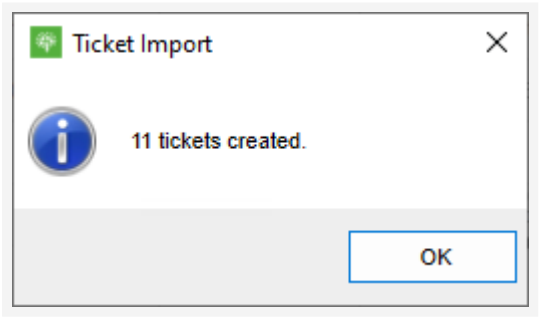


14 (Optional) Enter a new name for the file in the *File name* field.

15 Click **Import**.

Result: The **Ticket Import** dialog appears:

Ticket Import dialog

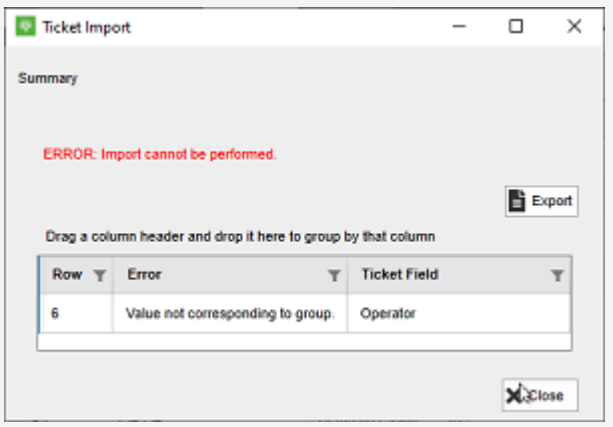


Result

The XLS file is imported and the tickets are added to the system.

ⓘ NOTE: If the import fails, an error dialog appears providing details about the issue. To fix it, go back to Step 7 (above) and correct the errors in the specified fields.

Error message in Ticket Import dialog



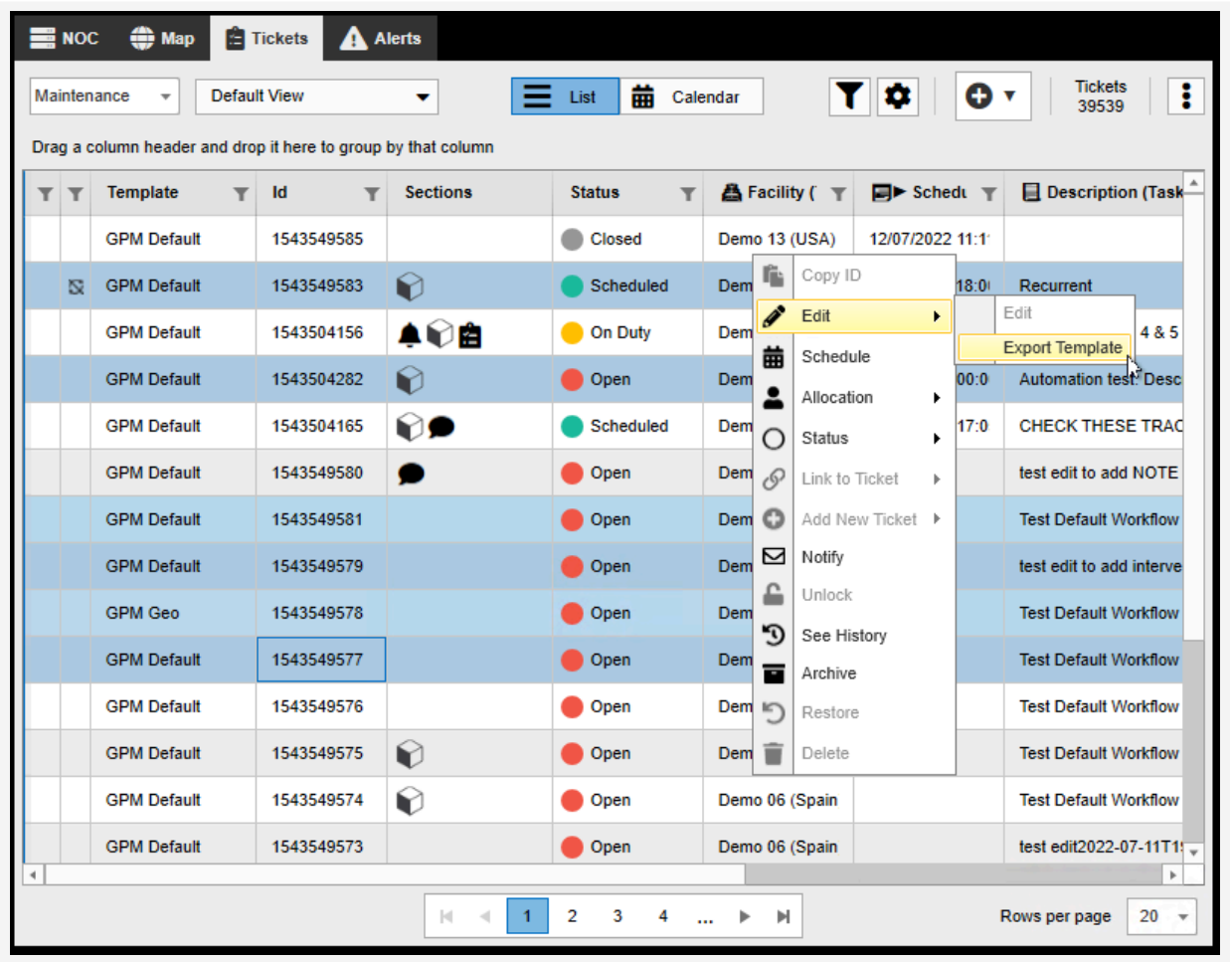
Edit tickets in bulk

Before you begin

This task requires an administrator password for security validation.

To edit tickets in bulk, follow these steps:

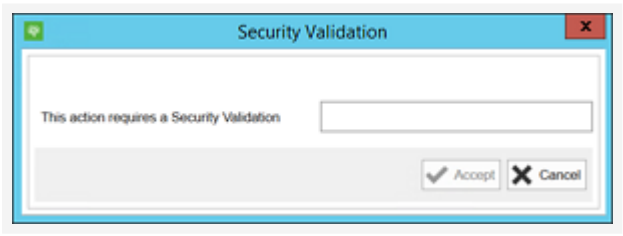
1. In the Tickets module, hold CTRL and click all the tickets you want to edit, then right-click the list to open the options menu, then select **Edit** and click **Export Template**.



> ☆ IMPORTANT: The tickets must belong to the same template (for example, "GPM Default").

Result: The **Security validation** dialog appears:

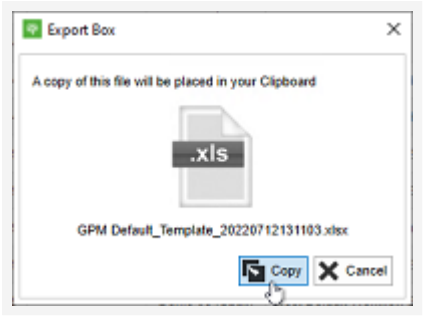
Security validation dialog



2. Enter the administrator password.

   Result: The **Export box** appears:

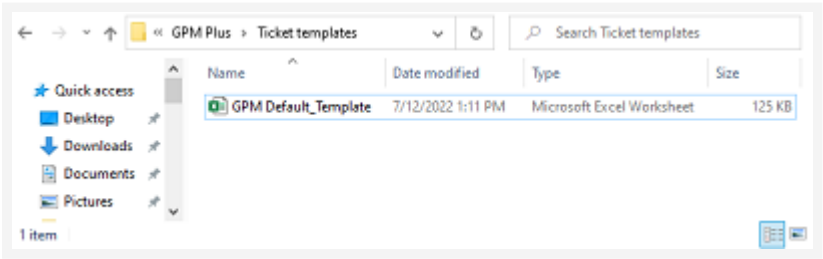
Export box



3. In your computer's File Explorer, navigate to the folder in which you want to save the XLS file and press CTRL+V, or right-click and select **Paste**.

   Result: The XLS file is copied to the folder you selected:

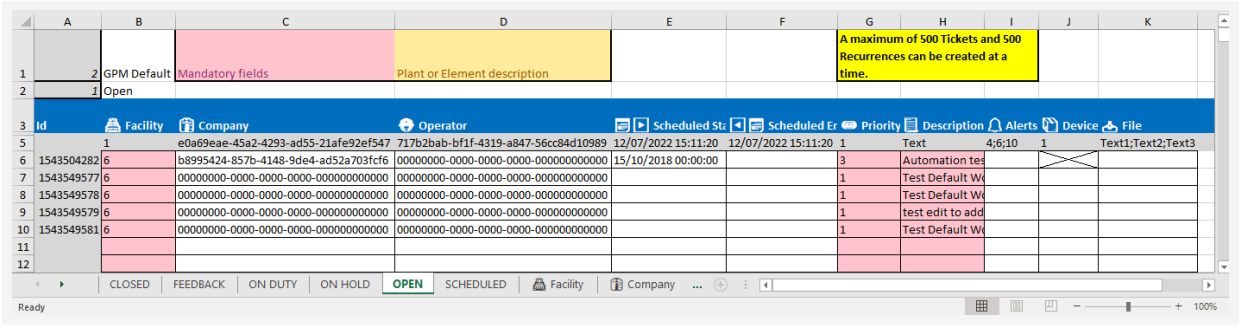
File Explorer



4. Open the XLS file, where each row corresponds to a ticket, and each column corresponds to a field.

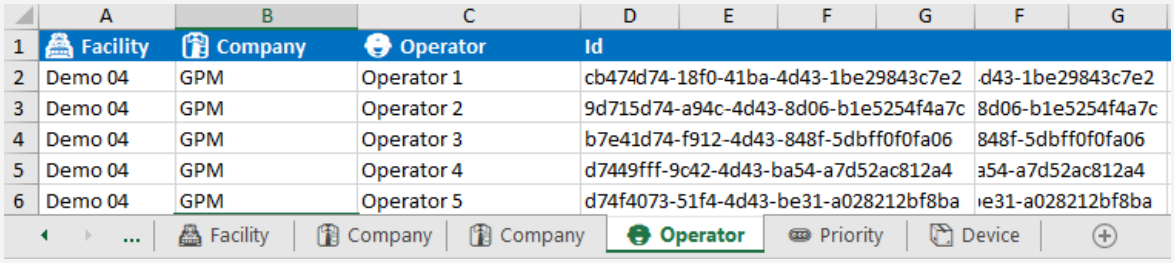
ⓘ NOTE: For more information, or to see the fields in context, see the Edit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticketEdit Ticket dialog in the instructions to create a maintenance ticket.

Ticket edition in Microsoft Excel



☆ IMPORTANT: The column headers display examples of the format for each field. Each page in the XLS file provides and ID and a reference for the available options in your system.

Sample IDs and references for the "Operator" field



5 Edit the cells corresponding to the fields you want to edit.

ⓘ TIP: For each field, you can go to the corresponding page in the XLS file, select and copy the relevant ID, and paste it in the column of the corresponding field.

a *Facility*: if your portfolio has more than one plant, enter the ID of the plant for which you want to create the ticket.

b (Optional) *Company*: enter the ID of the company that must resolve the issue related to the ticket.

c. (Optional) *Operator*: enter the ID of an operator to assign the ticket to them.

d. (Optional) *Scheduled Start*: enter the date and time on which the ticket starts, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).

e. (Optional) *Scheduled End*: enter the date and time on which the ticket ends, using the format displayed on the column header (for example, DD/MM/YY 00:00:00).

f. *Priority*: enter a level of priority (for example, 1).

g. *Description*: enter a description for the ticket.

h. (Optional) *Alarms*: enter the ID for the alarms to which you want to link the ticket.

i. (Optional) *Element*: enter the ID for the elements to which you want to link the ticket.

j. (Optional) *File*: enter the IDs of the files to which you want to link the ticket.

k. Repeat sub-steps (a) to (j) for each ticket you want to create.

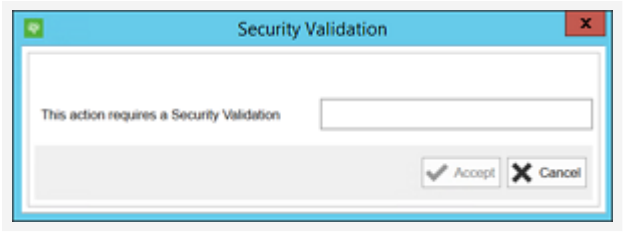
6. Press **CTRL+S**, or click on **File** and select **Save**.

Result: The changes to the XLS file are saved.

7. In the Tickets module of GPM SCADA, click the ⊕ icon and select **Create/Edit in bulk**.

Result: The **Security validation** dialog appears:

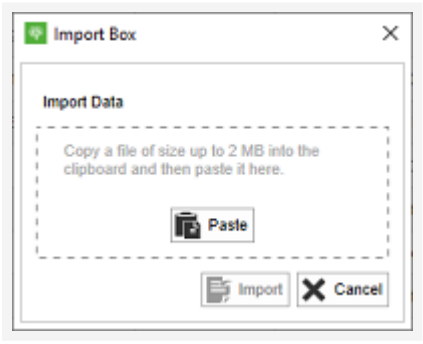
Security validation dialog



8. Enter the administrator password.

Result: The **Import box** appears:

Import box



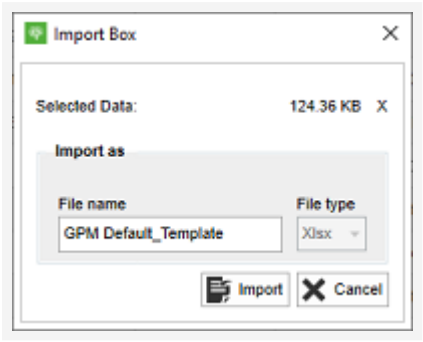
9. In your computer's File Explorer, select the XLS file with the ticket template and press **CTRL+S**, or right-click the XLS file and select **Copy**.

   Result: The XLS file is copied to your clipboard.

10. In the Import box, click **Paste**.

    Result: The XLS file is copied to the Import box.

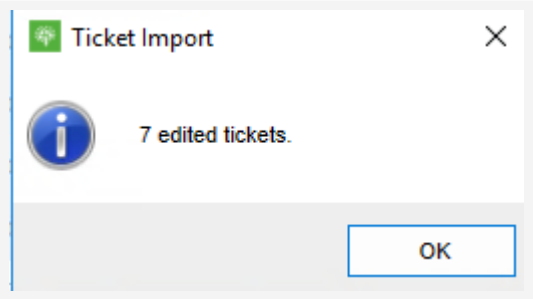
Import box



11. (Optional) Enter a new name for the file in the *File name* field.
12. Click **Import**.

Result: The **Ticket Import** dialog appears:

Ticket Import dialog

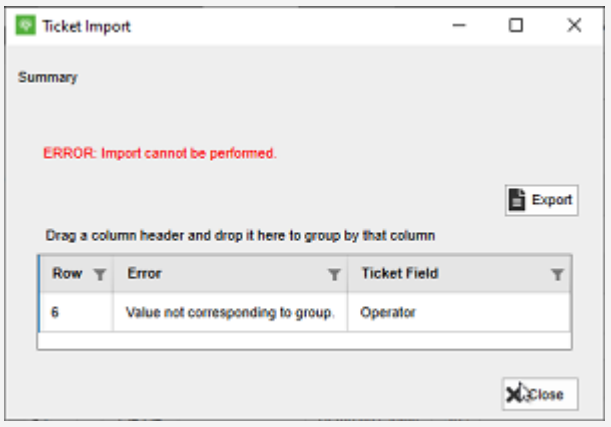


Result

The XLS file is imported and the tickets are added to the system.

ⓘ NOTE: If the import fails, an error dialog appears providing details about the issue. To fix it, go back to Step 7 (above) and correct the errors in the specified fields.

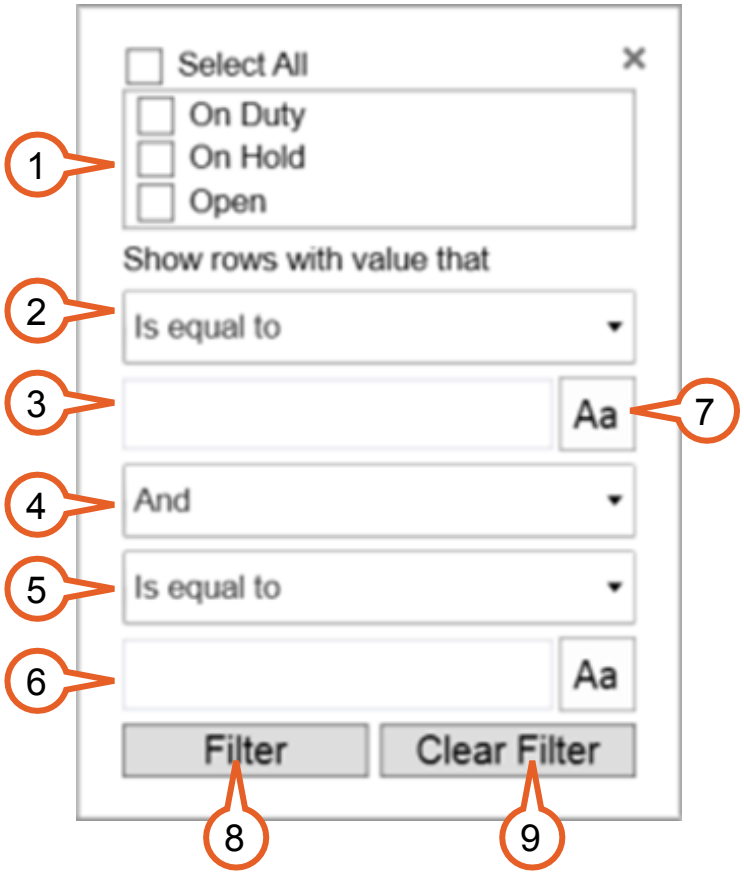
Error message in Ticket Import dialog



Advanced filters

Advanced filters allow you to filter the content of specific columns in a table using two mechanisms. The first mechanism allows you to select the values that you want to display from a list containing all the values of a column. The second mechanism allows you to apply a simple Boolean expression with two conditions to filter the values of a column.

Advanced filter



1. **Available values**
2. **First Boolean condition**
3. *First values*
4. **Boolean operator**
5. **Second Boolean condition**
6. *Second values*
7. **Case-sensitive button**
8. **Apply filter**
9. **Clear filtering criteria**

Use advanced filters

To use advanced filters, follow these steps:

1. Click the icon on any column where the advanced filtering option is available.
   Result: The **Advanced Filtering** dialog appears.
2. (Optional) Select the values that you want to display on the table.

   > ⓘ NOTE: Selecting values automatically displays them on the table.

3. In the **Show rows with value that** section, select the first Boolean condition from the drop-down list.
4. In the first *Values* input field, enter the first values.

   (Optional) Toggle the Aa button to turn the case-sensitive option on or off.

   > ⓘ NOTE: The case-sensitive button is not available when you are filtering numerical values.

5. Click the second **Boolean operator** drop-down menu and select the second Boolean condition from the drop-down list.
6. In the second *Values* input field, enter the second values.

   (Optional) Toggle the Aa button to turn the case-sensitive option on or off.

   > ⓘ NOTE: The case-sensitive button is not available when you are filtering numerical values.

7. Click **Filter**.

Result

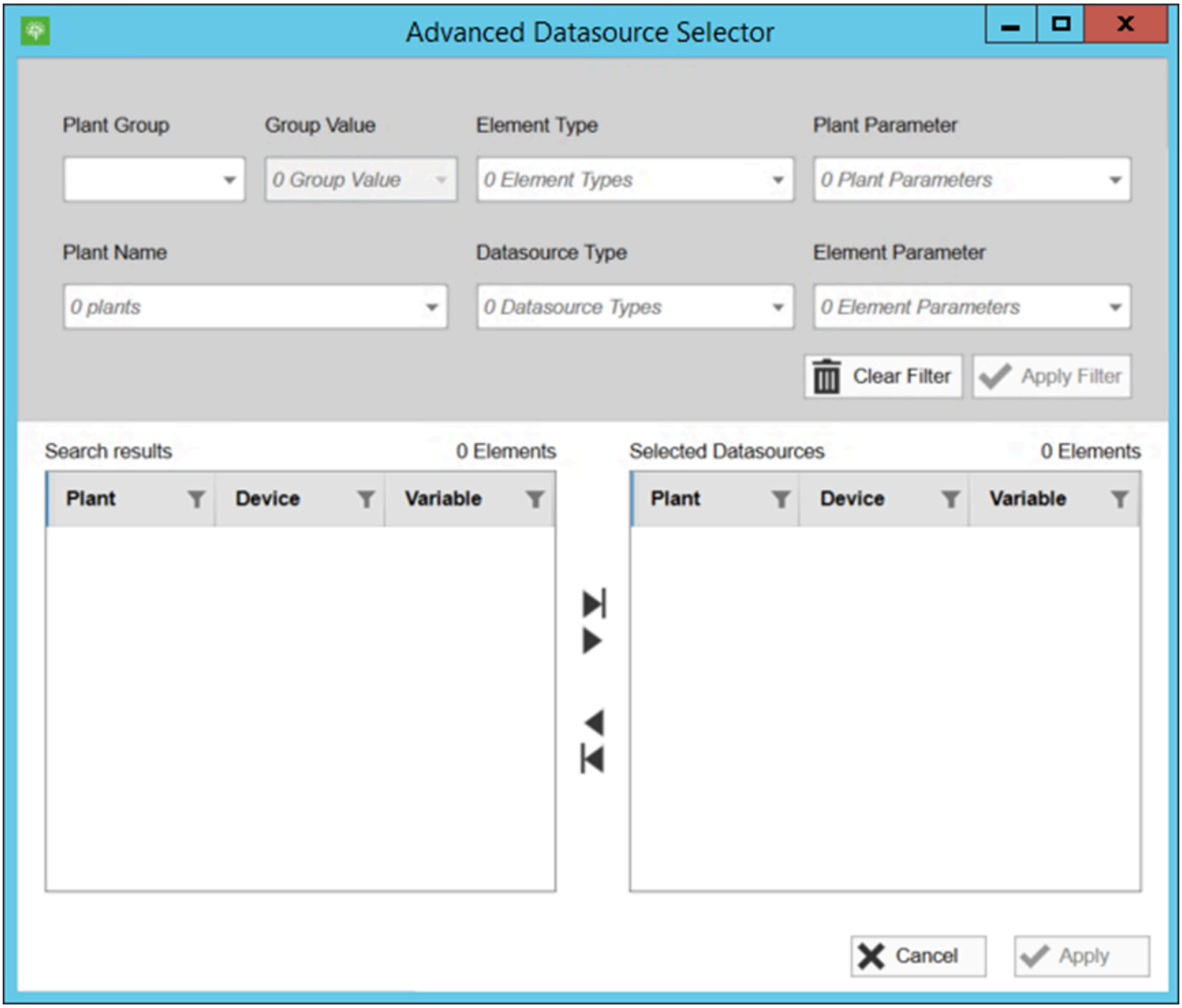
The filter is applied and elements matching the filtering criteria are displayed on the table. The icon on the column turns blue.

> ⓘ NOTE: You can clear all the criteria by clicking **Clear Filter**.

Advanced Datasource Selector

The Advanced Datasource Selector allows you to select source parameters and add them to queries. You may also export data to a file or import a file to process its data.

Advanced Datasource Selector



Add parameters to queries

To add parameters to a query using the Advanced Datasource Selector, follow these steps:

1. In the **Plant Group** drop-down menu, select the group to which the plant belongs.
2. In the **Group Value** drop-down menu, select one or more values.
3. In the **Element Type** drop-down menu, select one or more types of element.
4. In the **Plant Parameter** drop-down menu, select one or more parameters.
5. In the **Plant Name** drop-down menu, select one or more plants.
6. In the **Datasource Type** drop-down menu, select one or more types of datasource.
7. In the **Element Parameter** drop-down menu, select one or more parameters.
8. Click **Apply filter**.

   > ⓘ NOTE: You can clear a filter that was already applied by clicking **Clear Filter**.

9. In the **Search Results** panel, click to select the parameters, then click the ▶ icon to move them to the **Selected Datasources** panel.
10. Click **Apply**.

Result

The selected parameters are added to your query.

Import and export data

Some areas of the user interface allow you to use the Import and the Export features.

Importing data from files lets you add configurations or process external data. Depending on the user interface, the import icon may appear as ⊕ or as ✓.

Exporting data allows you to create files in the most common formats to analyze it outside of GPM SCADA. Depending on the user interface, the export icon may appear as 📝 or as 🔔.

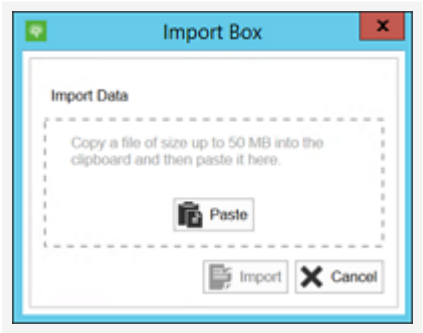
Import data from a file

To import data from a file to GPM SCADA, follow these steps:

1. Click the ⊕ or the ✓ icon.

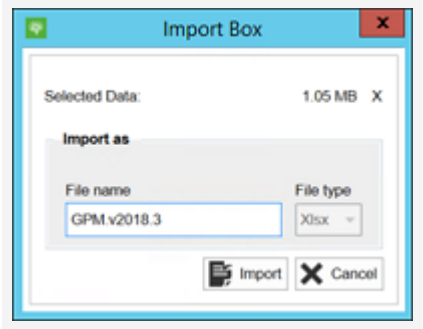
   Result: The **Import Box** dialog opens:

   ### Import Box

2. On your local machine, copy the file tht you want to import to your clipboard.
3. On the **Import Box** dialog, click **Paste**.
4. Review the information and click **Import**.

   ### Import Box with file

Result

The file is imported to GPM SCADA.

Export data to a file

To export data to a file, follow these steps:

1. Click the or the icon.

   Result: The **Export Box** dialog opens:

   Export Box

2. (Optional) If exporting to multiple formats is enabled, select the export format:
   - **Clipboard**: copy data to your clipboard as text.
   - **Data file**: export the data to a Microsoft Excel format.
   - **JPG image:** export the data as a JPEG image.
3. On the **Export Box** dialog, click one of the following options:
   - **Open**: open the data in the in-app viewer.
   - **Copy**: copy the exported file to your clipboard.
   - **Cancel:** cancel the export operation.
4. If you selected **Copy**, paste the copied file to a location on your machine.

Result

The file is exported to your local machine in the format of your choice.

GPM Ticket Manager User Guide

Welcome!

Welcome to the Technical Documentation space for GPM Ticket Manager.

Here you can download all the official documentation that we create to help you use our software. If you have any questions or feedback, please contact help@greenpowermonitor.com.

Latest release

- Current release: version 2022.3
  - User Guide: Version 2022.3

About this guide

Disclaimer

The content of this document is not representative of every product configuration. Each product instance is configured to meet the needs of the intended users, who may or may not require certain features and options. Therefore, the features and options covered in this document may differ from those available on your product configuration.

Conventions

Typographical conventions

UI control	User interface controls, commands, and keywords in body text.
Blue-bold	Option in a menu.
Input field	Input field where user input is expected.
Underline	Link to another section of the guide.
`monospace`	Snippets of code that the user must input or can use as a reference.
<...>	Generic parameters that must be replaced by specific code or text.
[...]	Generic parameters that are replaced by dynamic text.

Notes and recommendations

ⓘ NOTE: Designates a note or reference related to the surrounding text.

BEST PRACTICE: Designates a usage recommendation related to the surrounding text.

⚠ CAUTION: Designates a warning or alert related to the surrounding text. You should exercise caution to avoid an undesirable outcome.

Before you begin

> ⓘ NOTE: The GPM Ticket Manager App is only available on Android devices.

To work with the GPM Ticket Manager, your GPM Plus system must be configured to link with the app. You must also ensure that your user account has an Operator profile, and that the tickets are correctly configured.

Prerequisites

- User account linked to an Operator profile in the GPM Plus system. For more information, see the instructions to link a user account to an Operator profile.
- Active GPM Ticket Manager key in your GPM Plus server.

  > ⓘ NOTE: To configure GPM Plus to support linking with the GPM Ticket Manager, contact your GPM representative.

- Tickets configured with the template for the GPM Ticket Manager. For more information, see the instructions to configure the template for tickets.

Configure tickets for the GPM Ticket Manager app

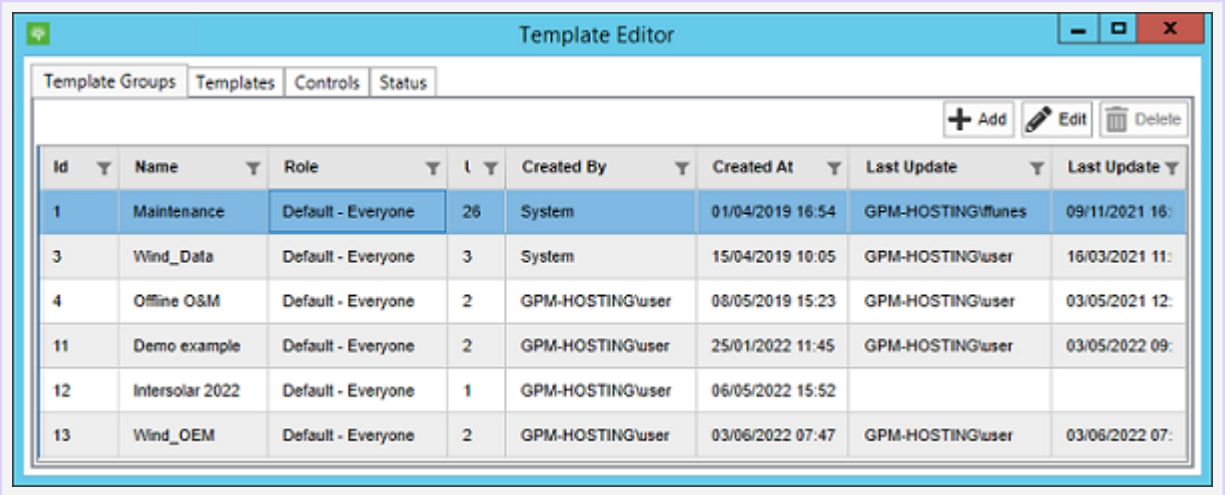
Before you begin

This task requires an administrator password.

To configure a ticket template for use with the GPM Ticket Manager, follow these steps in GPM Plus:

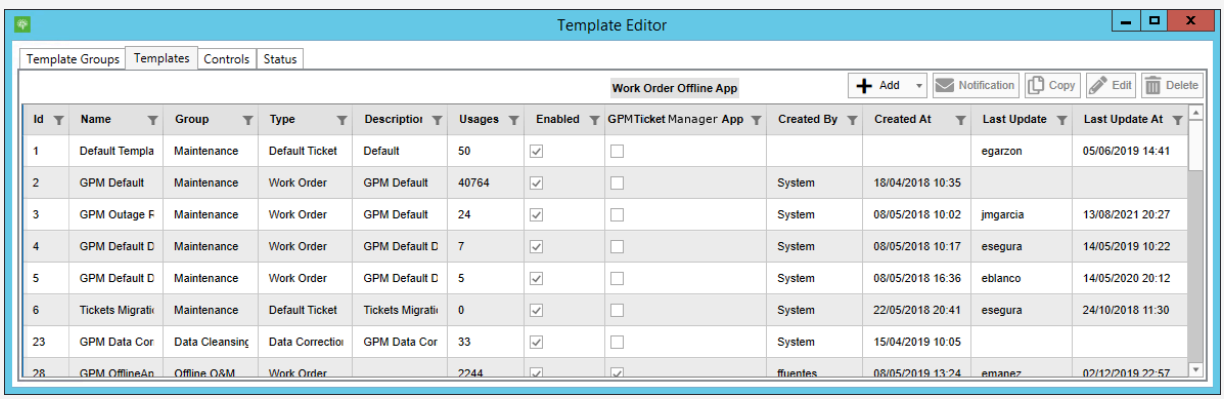
1. Click the ⚙ icon in the top navigation bar to open the **Setup** screen and select **Tickets configuration** to open the Template Editor screen:

Template Editor



2. Select the **Templates** tab.

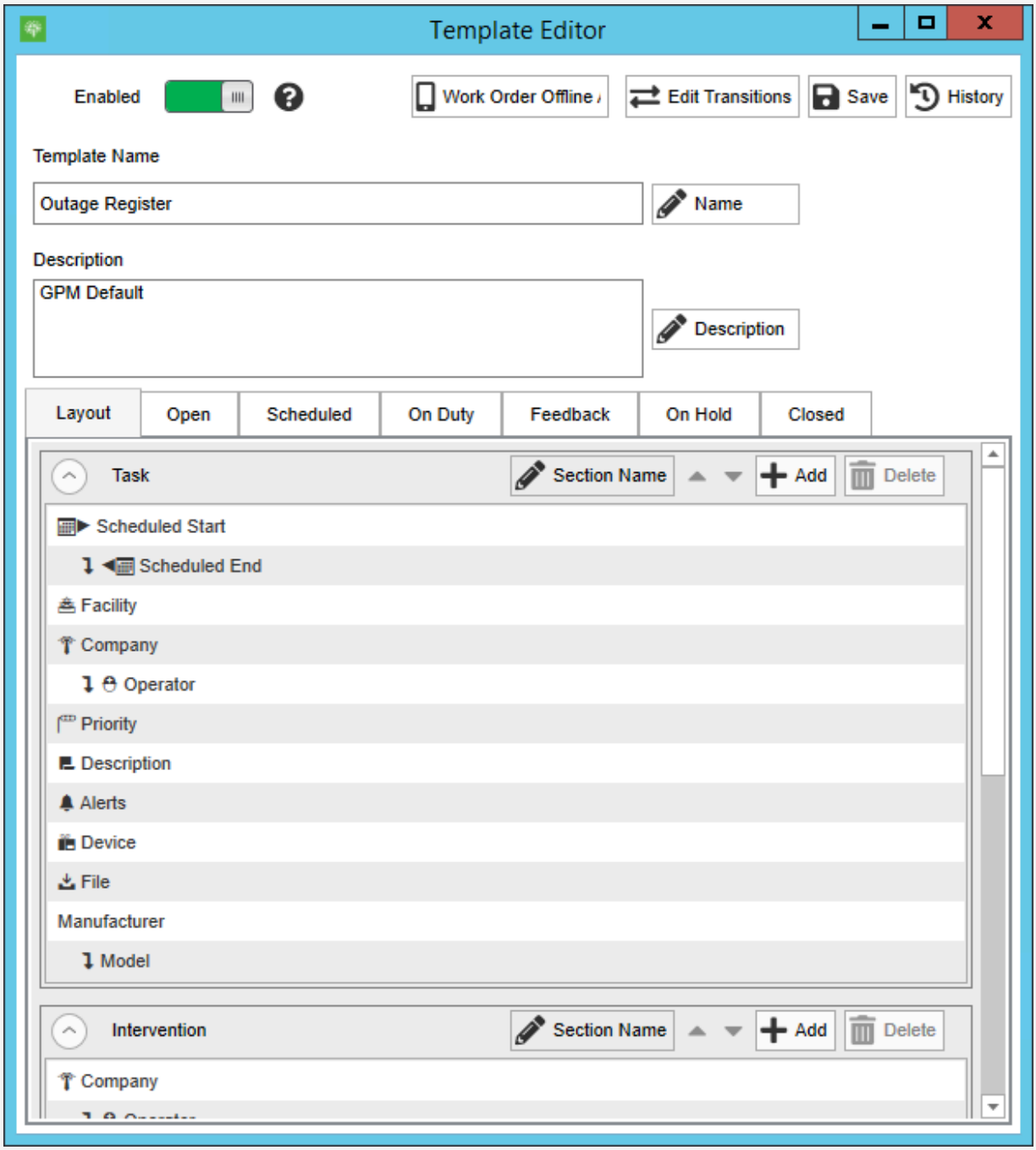
Templates tab



3 Select the template you want to make available on the GPM Ticket Manager, then click **Edit**.

Result: The Template Editor dialog appears:

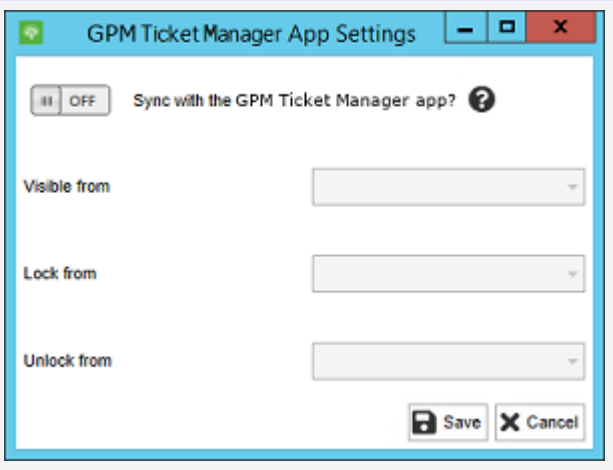
Template Editor



4 Click **GPM Ticket Manager**.

Result: The GPM Ticket Manager settings menu appears:

GPM Ticket Manager settings



5. Toggle on **Sync with the GPM Ticket Manager app?**
   Result: The drop-down menus become available.
6. Open the **Visible from** drop-down menu and select the status from which the tickets become visible on the app.
7. Open the **Lock from** drop-down menu and select the status from which the tickets become locked in GPM Plus.
8. Open the **Unlock from** drop-down menu and select the status from which the tickets become unlocked in GPM Plus.
9. Click **Save**.

Result

The changes are saved, and tickets from the selected template become available to operators using the GPM Ticket Manager.

Get started with GPM Ticket Manager

To use the GPM Ticket Manager, you must first download the app from the Google Play Store, then sync it with your GPM Plus system and log in using your credentials.

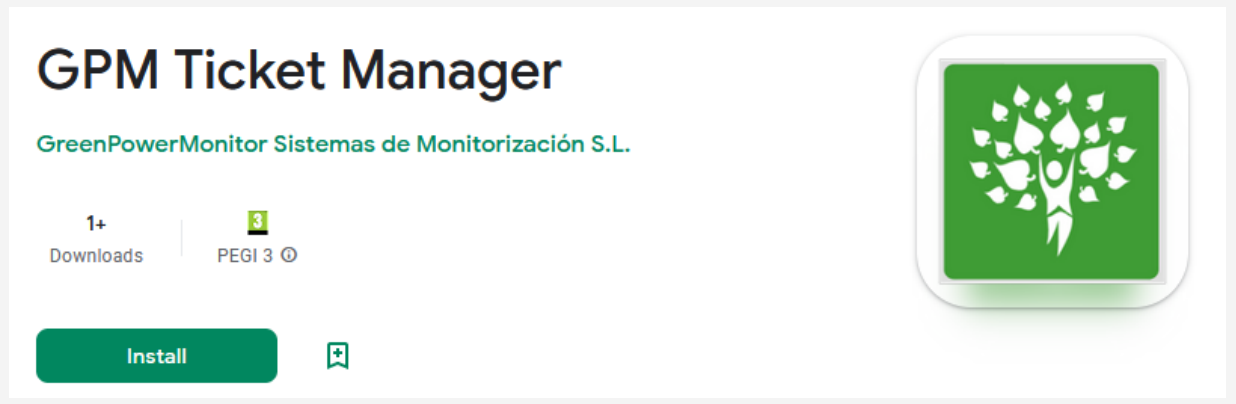
Download the GPM Ticket Manager

To download the GPM Ticket Manager to your mobile device from the Google Play Store, follow these steps:

> ⓘ NOTE: These instructions relate to a third-party platform that does not depend on GreenPowerMonitor. All information is to the best of our knowledge, and may be subject to change.

1. In your web browser, go to
   `https://play.google.com/store/apps/details?id=com.greenpowermonitor.TicketsOfflineApp`
   .
   Result:

   Download page

2. Click **Install**.

Result

The GPM Ticket Manager is downloaded to your device.

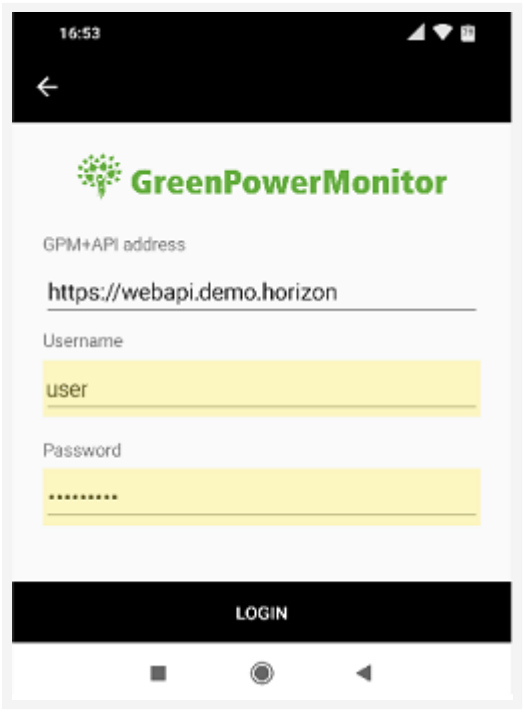
Log in and sync with GPM Plus

To log into the GPM Ticket Manager and sync the app with GPM Plus, follow these steps:

1. Open the GPM Ticket Manager app.

   Result:

   GPM Ticket Manager

2. Enter the *GPM+API address* provided by your system administrator.
3. Enter your *User name*.
4. Enter your *Password*.
5. Click **Log in**.

Result

The app is linked to the GPM Plus system and you are logged in.

Manage tickets

The Inbox of the GPM Ticket Manager displays all the tickets that have been assigned to you. From here, you can view the information and download tickets to work with them.

> ⓘ NOTE: Downloaded tickets become blocked in the GPM Plus, where the icon appears next to them to notify users that someone is working on them offline. When the operator goes back online and updates the ticket, it becomes unblocked. Administrators can unlock tickets if needed (for example, if the operator's mobile device is lost); this action is logged over the ticket ID.

You can make the following changes to a ticket:

- Add interventions
- Add notes
- Change status

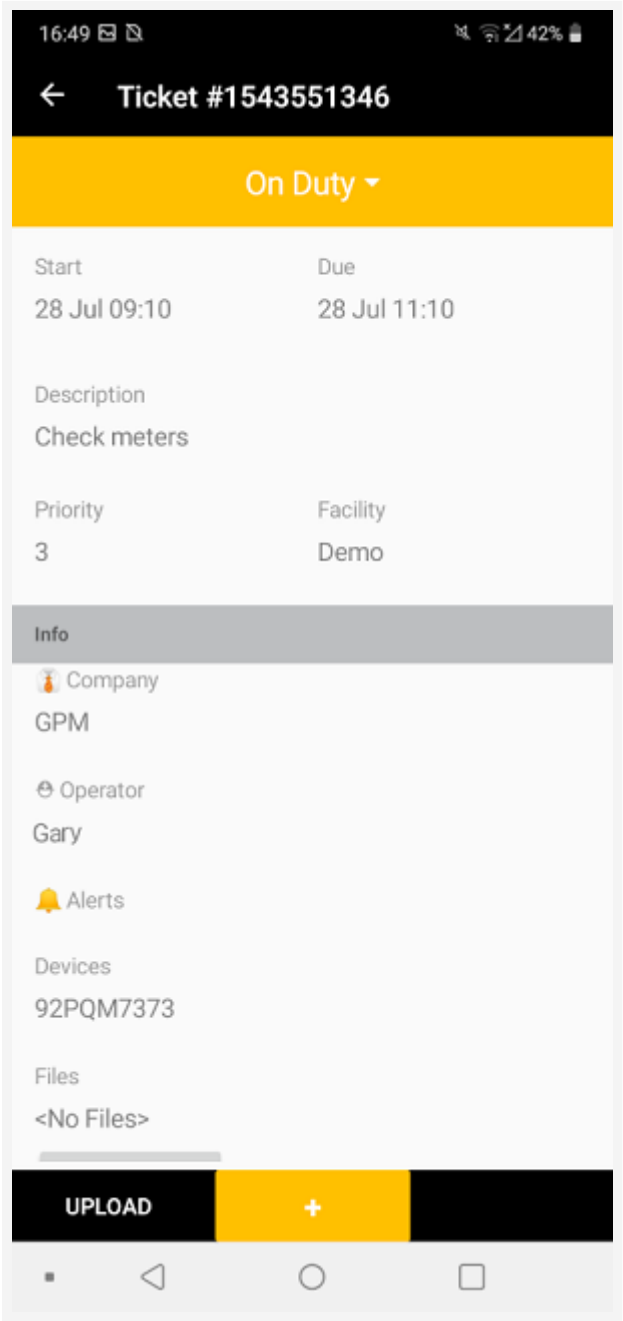
When you finish working with a ticket, you can upload it to the system.

Add interventions to tickets

To add an intervention to a ticket, follow these steps:

1. In the Inbox of the GPM Ticket Manager, select a downloaded ticket to open it.
   Result: The ticket opens.

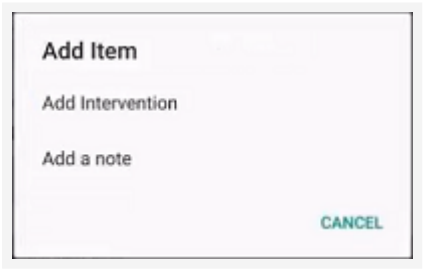
   Ticket



2. Click the **+** icon.

Result: The **Add item** dialog appears:

Add item dialog

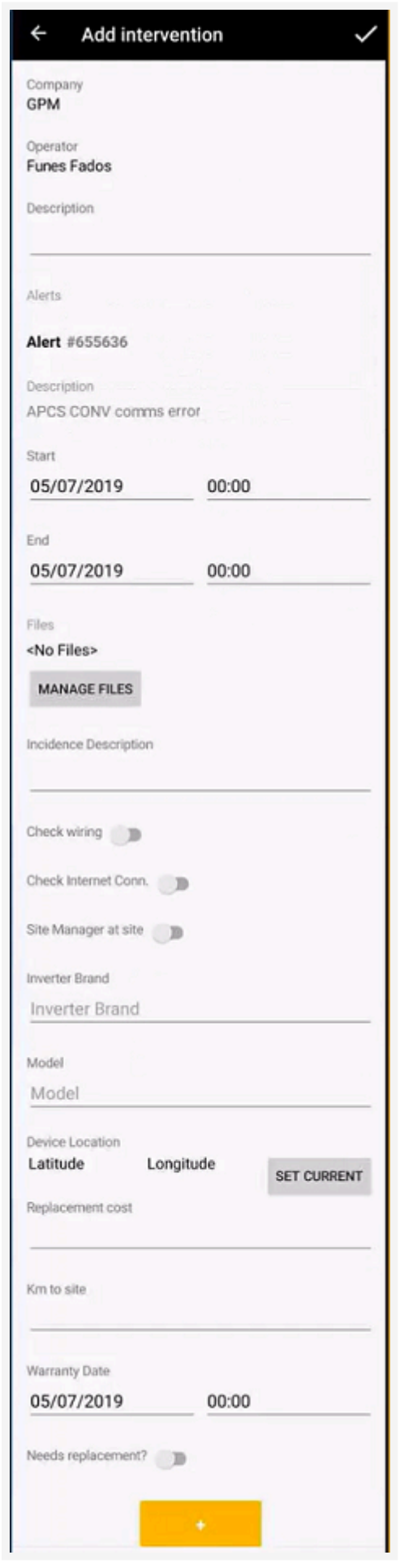


3 Click **Add intervention**.

Result: The **Add intervention** screen appears:

> ⓘ NOTE: The fields available in this screen depend on the configuration of the ticket template in GPM Plus.

Add intervention screen



4. Enter the information for the intervention:
   a. Enter a *Description* (for example, "Replaced wire").
   b. In the *Start* fields, enter the date and time at which the intervention began.
   c. In the *End* fields, enter the date and time at which the intervention ended.
   d. (Optional) Click **Manage files** to add files to the intervention.
   e. Enter an *Incidence description* (for example, "Wire was damaged").
   f. Enter any other relevant information.
5. Click the ✓ icon.

Result

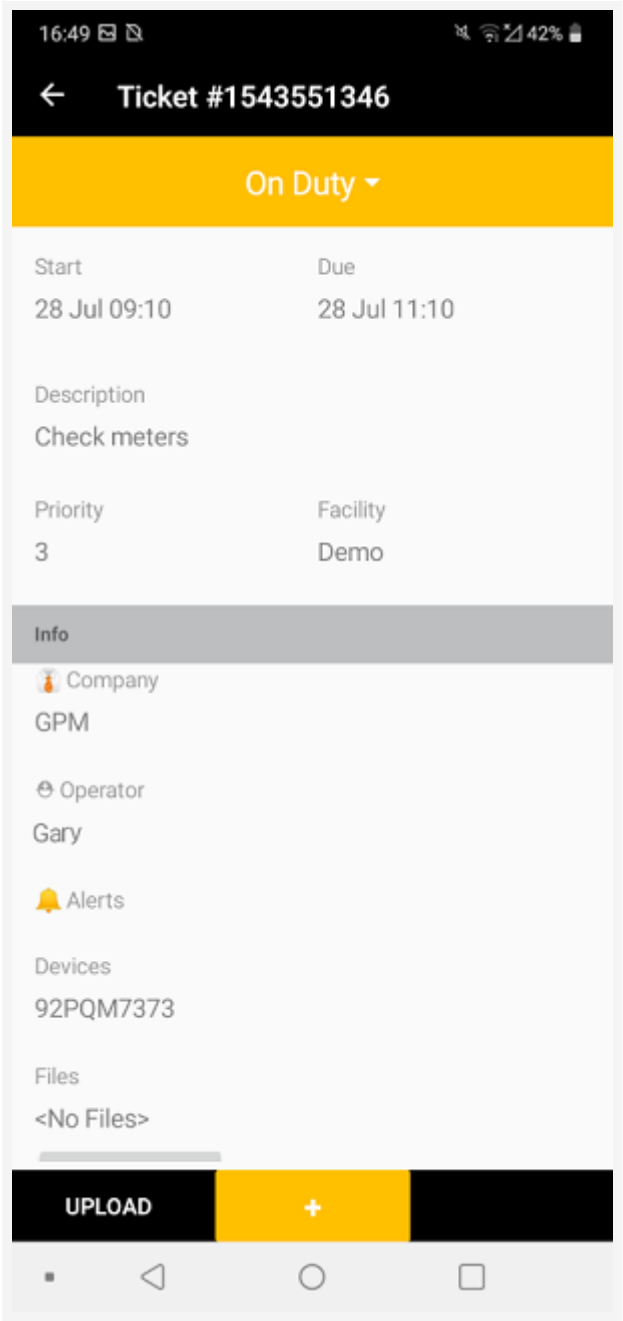
The intervention is added to the ticket. After you upload the ticket, your note appears in the "On field operation" tab of the ticket screen in GPM Plus.

Add notes to tickets

To add a note to a ticket, follow these steps:

1. In the Inbox of the GPM Ticket Manager, select a downloaded ticket to open it.
   Result: The ticket opens.

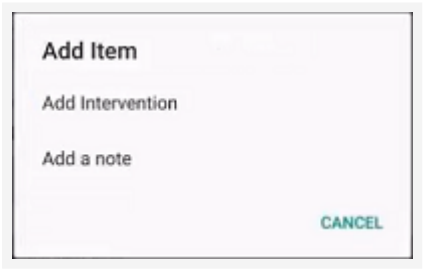
   Ticket



2. Click the **+** icon.

Result: The **Add item** dialog appears:

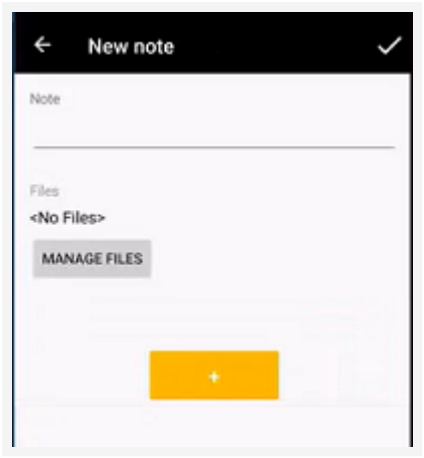
Add item dialog



3. Click **Add a note**.

   Result: The **New note** screen appears:

New note



4. Enter the text in the *Note* field.
5. (Optional) Click **Manage files** to upload files.
6. Click the ✓ icon.

Result

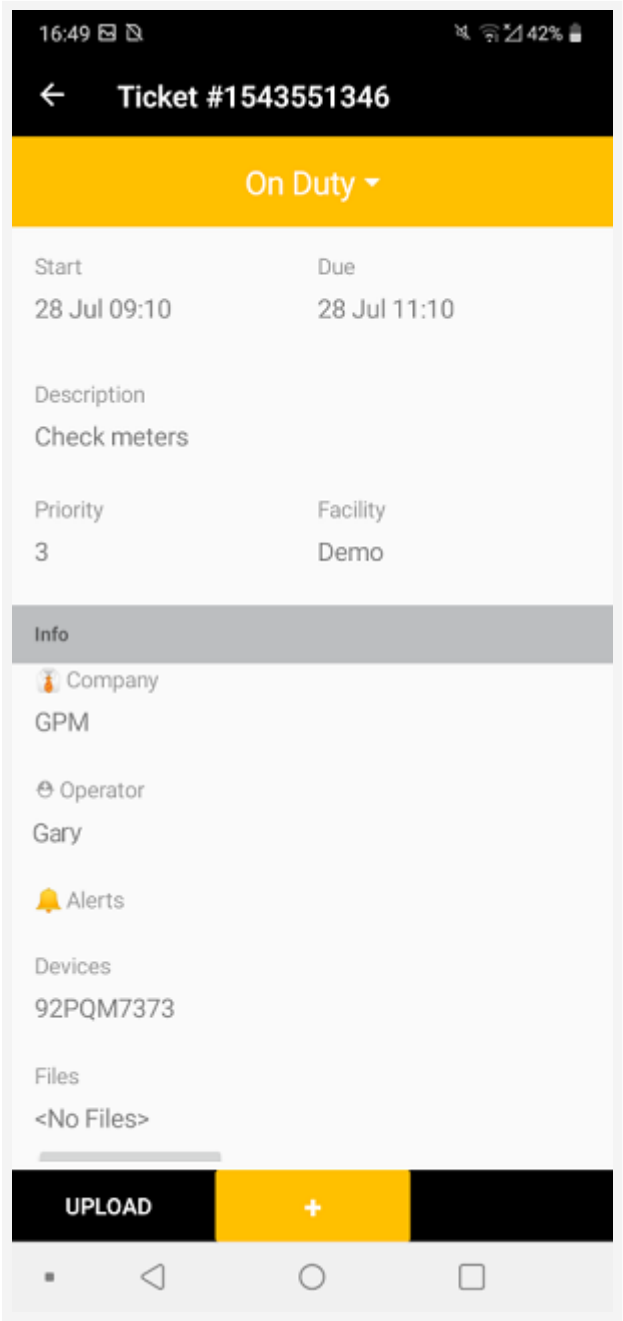
The note is added to the ticket. After you upload the ticket, your note appears in the "Conversation" tab of the ticket screen in GPM Plus.

Change ticket statuses

To change the status of a ticket, follow these steps:

1. In the Inbox of the GPM Ticket Manager, select a downloaded ticket to open it.
   Result: The ticket opens.

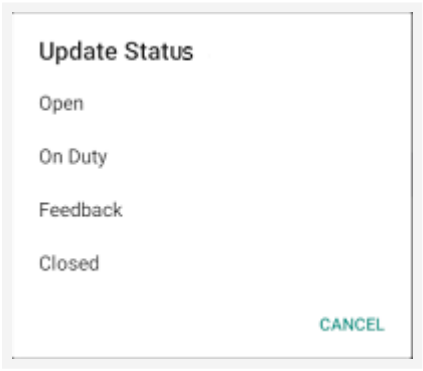
   Ticket



2. Click the status of the ticket.

Result: The **Update status** dialog appears:

Update status dialog



3 Select the status to which you want to set the ticket.

Result

The status is changed. After you upload the ticket, the status change is registered in GPM Plus.

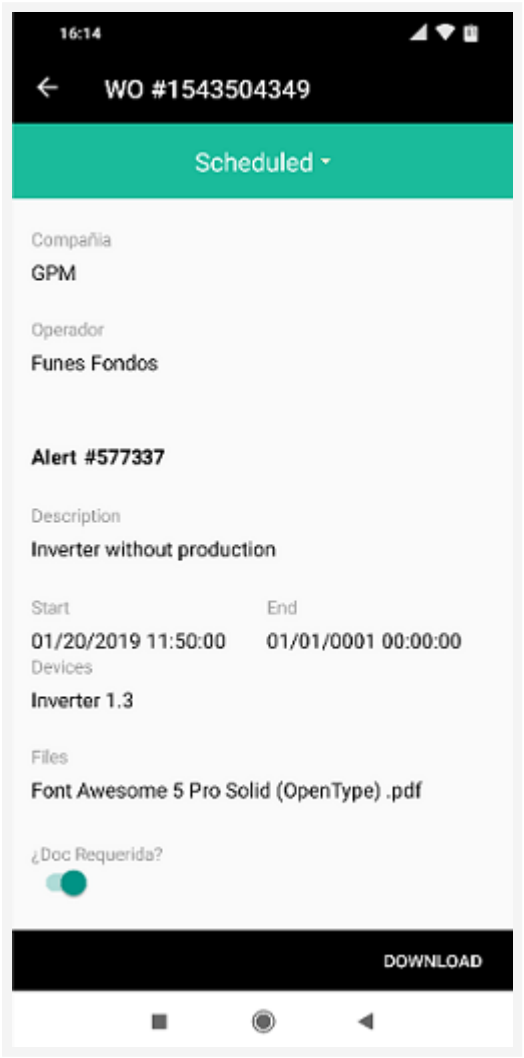
Download tickets

To download a ticket, follow these steps:

> ⓘ NOTE: Downloaded tickets become blocked in the GPM Plus, where the icon appears next to them to notify users that someone is working on them offline. When the operator goes back online and updates the ticket, it becomes unblocked. Administrators can unlock tickets if needed (for example, if the operator's mobile device is lost); this action is logged over the ticket ID.

1. In the Inbox of the GPM Ticket Manager, select the ticket to open it.
   Result: The ticket opens.

Ticket



2 Click **Download**.

Result: The **Download tickets** dialog appears:

Download tickets dialog

3 Click **Yes**.

Result

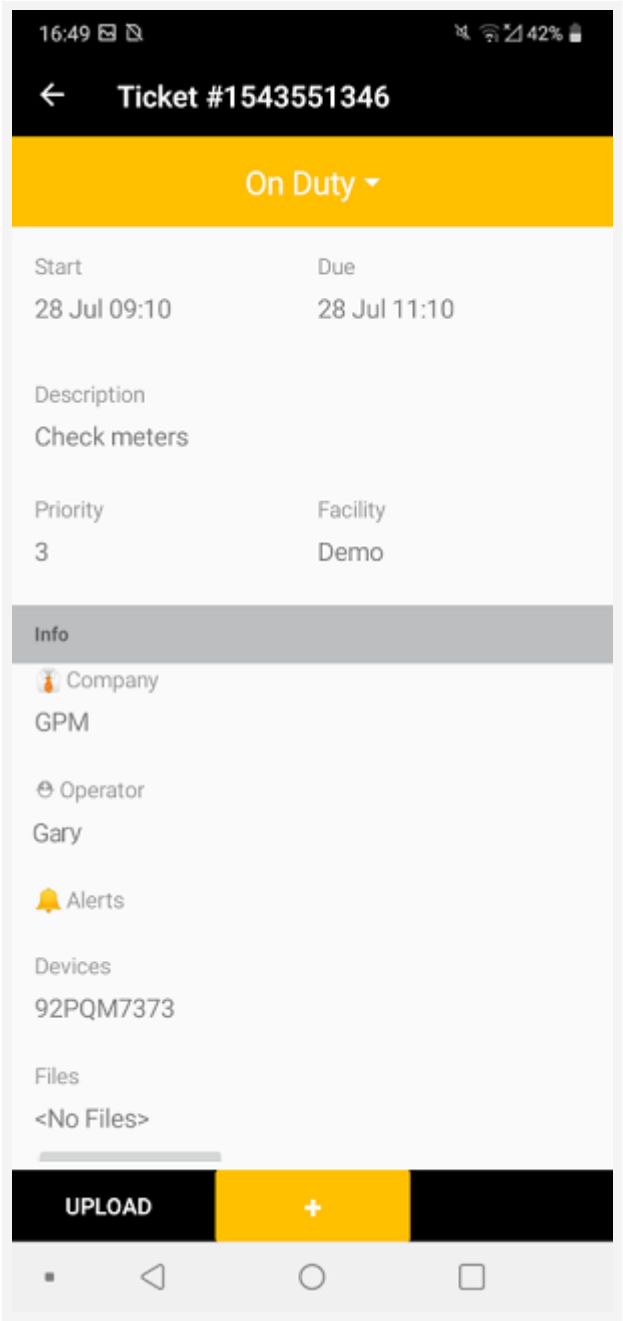
The ticket is downloaded and its status changes to "On Duty". It also appears as "Locked" in GPM Plus.

Upload tickets to GPM Plus

To upload a ticket to GPM Plus, follow these steps:

1. In the Inbox of the GPM Ticket Manager, select a downloaded ticket to open it.
   Result: The ticket opens.

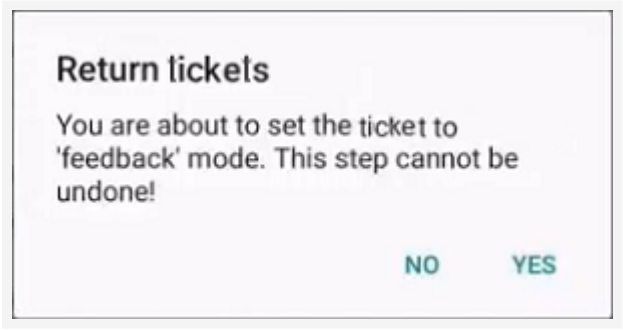
   Ticket



2. Click **Upload.**

Result: A confirmation dialog appears:

Return ticket dialog



3 Click **Yes**.

Result

The ticket is uploaded to GPM Plus and unlocked.