



Software Application Guide



Table of Contents

Accessing System Administration	5
Accessing System Administration	5
Logging In at the Job Site	6
About Configuring On-Site Verses from the Cloud	6
Logging In	6
Signing into the Project Cloud	8
Configuring Settings	9
Configuring Project Settings	9
Configuring Protocol Settings	13
Adding and Configuring Users	18
Editing Users	19
Configuring Roles	23
Configuring (Alarm Notification) Groups	25
Configuring Weather Settings	26
Searching User Action Logs	27
Configuring LAN/Ethernet Settings	28
Configuring Wi-Fi Settings	29
Configuring Cellular Settings	30
Configuring Date and Time Settings	31
Configuring Whitelist/Blacklist Settings	33
Configuring IP Tables	35
Configuring Proxy Settings	35
Configuring SSH Settings	36
Configuring Networks	37
Supported Network Protocols	37
Configuring a BACnet Network	37
Configuring a KMDigital Network	38
Configuring a Modbus Network	40
Configuring an SNMP Network	40
Configuring a Node-RED Network	42
Configuring Devices	43
Discovering Devices	43
Assigning Device Profiles	43
Editing a Device Profile	45
Editing Device Details	46
Creating a Site Topology	47
Virtual Point Program Examples	49
Finding and Deleting Orphan Nodes	53
Dashboards and Their Elements	54
About	54
Adding and Configuring Dashboards	54
Sharing Dashboards	57
Modifying (and Deleting) Dashboards	58
Creating and Adding Cards	59
Creating a Custom Card	59
Creating a KPI Card	61

Creating a KPI Gauge Card	63
Creating a Trend Card	65
Creating a Thermostat Card	66
Creating a Weather Card	68
Creating a Web Card	69
Creating a Text Editor Card	70
Creating a Report Card	71
Duplicating a Card Across Devices	72
Modifying Cards	73
Using Cards	75
Deleting a Card	79
Creating and Adding Decks	79
Modifying Decks	81
Using Decks	82
Deleting Decks	84
Adding a Canvas	84
Adding and Styling Canvas Points	85
Drawing and Configuring Canvas Zones	88
Adding a Report Module	92
Modifying Report Modules	92
Managing Schedules and Events	94
About	94
Know Before Beginning	94
Adding Schedules	94
Creating Events	96
Modifying and Deleting Schedules	97
Modifying and Deleting Events	97
Managing a BACnet Schedule Object	98
Managing Trends	102
Video Overview	102
About Trends	102
Creating Trend Graphs	102
Using the Binning Interval	104
Viewing Trends	107
Modifying and Deleting Trend Graphs	109
Exporting Trend Data	110
Managing Alarms	112
Video Overview	112
Know Before Beginning	112
Configuring a Point Value Alarm	112
Configuring a Gateway Status Alarm	116
Configuring a Network Status Alarm	117
Configuring a Device Status Alarm	118
Configuring a Point Status Alarm	119
Duplicating Alarms	120
Modifying and Deleting Alarms	121
Finding, Viewing, and Acknowledging Alarms	122
Managing Reports	124
About	124
Reports Overview	124

Important to Know	124
Configuring an All Points Report	125
Configuring an All Points By Device Report	126
Configuring an Energy Usage Report	127
Configuring a Manual Override Report	130
Configuring an Out of Range Report	131
Configuring a Runtime Report	133
Running a Saved Report Setting	134
Modifying and Deleting a Saved Report Setting	135
Linking to a Report	135
Using a Report	136
Changing Personal Profile Settings	138
Changing Your Personal Information	138
Configuring SMS Notifications	138
Changing Your Password	138
Accessing Your API Key	139
Configuring Your Offline Device Notifications	139
Switching Between Projects	140
Add-ons (and Data Explorer)	141
What are Add-ons?	141
Using Data Explorer	141
Node-RED Add-on	145
API Integration	146
Accessing Your API Key	146
Information	146
Troubleshooting	147
Restarting the Gateway	162
Gateway Restart Options	162
Software Updates	163
About Software Updates	163
Accessing Other Documents	164
How to Access Other Documents	164
Contact Information	166
Important Notices	167

Accessing System Administration

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Accessing System Administration

After KMC Customer Service has set up your company, a designated administrator will be emailed license credentials to sign into the project Cloud. An administrator can manage user, group, customer project, and box (KMC Commander IoT gateway) information.

To sign in the first time as an administrator, click the link in the email sent by KMC Controls at the initial purchase of KMC Commander.

For more details, see the *KMC Commander (Cloud) System Administration Application Guide* (see [Accessing Other Documents on page 164](#)).

Logging In at the Job Site

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Configuring On-Site Verses from the Cloud

Dashboards, schedules, trends, and alarms can be configured later from the Cloud as desired, but the following are the minimum functions to perform on-site (or performed as local via VPN):

- Configure **Settings**  (especially local-only settings). (See [Configuring Settings on page 9](#).)

Note: Cloud settings do not include these local-only settings: **Network Interfaces** (Ethernet, Wi-Fi, and Cellular), **Date & Time**, **Whitelist/Blacklist**, **IP Tables**, **Proxy**, and **SSH** settings), but those settings *can* be configured through a VPN.

- Recommended: Discover all known network devices and points (in Network Explorer) and set up profiles. (See [Configuring Networks on page 37](#), [Discovering Devices on page 43](#) and [Assigning Device Profiles on page 43](#).) See "Configuring Networks", "Discovering Devices", and "Assigning Device Profiles" in the *KMC Commander Software Application Guide*. (See [Accessing Other Documents on page 164](#)).

Note: The Cloud can discover devices and points. However, discovery of devices and points on site will be helpful if network troubleshooting is required.

Logging In

Before Internet is established

Before an Internet connection is established for the gateway (see [Configuring Network Interfaces](#)), sign in using Wi-Fi:

1. In a (Google Chrome or Safari) browser window, sign into KMC Commander using Wi-Fi (see [Connecting Wi-Fi and Making Initial Login](#)).
2. Enter your (case-sensitive) user **Email** and **Password**, as previously set up by a system administrator. (See [Accessing System Administration on page 5](#).)

Note: If you forget your password, select **Forgot password**, enter your email address, and you will receive an email with a link for resetting your password.

3. Choose the relevant **License** (if more than one is available to you).

Note: If the correct license is not available, see [License and Project Problems on page 154](#).

4. **Select Submit.**

Note: **Networks Explorer**  will appear.

Configure the settings as needed.

After Internet is established

After an Internet connection is established for the gateway (see [Configuring Network Interfaces](#)), sign into the project Cloud at app.kmccommander.com. (See *Signing into the Project Cloud* on page 8.)

Signing into the Project Cloud

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

After an Internet connection has been established for the gateway (see [Configuring Network Interfaces](#)), signing into projects through the project Cloud is almost always recommended and can be done remotely.

1. Enter app.kmccommander.com in a web browser.

Note: Chrome or Safari are recommended.

2. Enter your KMC Commander Project Cloud Login email and password.
3. Select **Login**.

Note: For optional [Google Single Sign On](#), Google credentials can be used for login if the Gmail credentials are entered as a new User in the System Administration (see [Accessing System Administration on page 5](#)).

4. Select your project from the drop-down list (if more than one).

Note: Project options are shown as Project Name (license name for the KMC CommanderIoT gateway). Multiple gateways may be part of a single project, such as in “My Big Project (IoT Box #1)”, “My Big Project (IoT Box #2)”, and “My Big Project (IoT Box #3).”

Note: A Google map with red pins can show the location of projects if the addresses are entered in the (Cloud) KMC license administration. (To use this feature, supply KMC Controls with your desired project address information for the license server.) Select a red pin, then **Click to Continue** to open that project.

Note: During initial setup, the (Internet) network connection must have a DHCP server to get an address, and the PC being used must be set to have a dynamic IP address rather than a static address.

Note: It may take several minutes before all cards and present values are visible.

Note: The cards that are viewable depend on the user’s access profile.

Note: The Settings section (gear icon) in the Cloud has fewer options than when connecting to the local gateway. (See [Configuring Settings on page 9](#).)

Note: In a Cloud dashboard, cards can show points from devices from multiple KMC Commander (IoT gateway hardware) boxes if multiple boxes exist in the project.

Configuring Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Note: For your personal profile settings, see [Changing Personal Profile Settings](#) on page 138.

Configuring Project Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Accessing Project Settings

Go to **Settings** , then **Project**.

Under the Project Settings header

The name and time zone of the project (as set in the KMC Commander license server) shows here.

Auto Archive Alarms

1. Choose whether or not to auto archive alarms. If you choose **On**:
 - Alarms that are acknowledged in Alarm Manager will be archived after the number of hours (1 minimum) entered in **Acknowledged and Older Than (Hours)**.
 - All alarms, whether acknowledged or not, will be archived after the number of days (1 minimum) entered in **Any Alarm Older Than (Days)**.
 - Archived alarms can be hidden or viewed. (See [Finding, Viewing, and Acknowledging Alarms](#) on page 122.)
2. Select **Save**.

Dashboard

Point ID Column from Card Detail

1. Choose to either **Show** or **Hide** the **Point ID** column from the backs of cards on dashboards.
2. Select **Save**.

Dashboard Deck Mode

1. From the dropdown menu, select the default view mode for decks on dashboards.

Note: Individual decks can be changed from the default to another view mode (see [Switching Between Deck View Modes](#) on page 83) However, whenever a dashboard reloads, the decks will return to this default. Also, when you add a deck to a dashboard it will appear in this view mode.

2. Select **Save**.

Read Time After Point Writes (Seconds)

The value entered here is the seconds interval after the system writes a point that it will read the new value.

Note: Typically the system writes to a point within half a minute (depending on network speed and other factors), but a read confirmation of the successful write (e.g., a setpoint displayed on a card changes from the old value to the new value) may take several minutes. If errors are occurring when reading, adding an extra time interval may help reduce errors.

1. If desired, enter a custom interval (in seconds).
2. Select **Save**.

Display Point Override

1. Choose whether an indication should display on cards that a point is in override. If you choose **On**:
 - A border (together with a hand icon), colored the [Point Override Color on page 10](#), will appear around the slot of an overridden point.
 - Hovering over the point's name will cause information about the override to appear.

Note: The override indication will display when a point's value is written at the same or higher priority than the [Default Manual Write Priority on page 15](#) setting, found in **Settings**  **> Protocols**.

2. Select **Save**.

Point Override Color

1. If [Display Point Override on page 10](#) is **On**, do *one* of the following to choose a color for the override indication:
 - Select a color, using the color selector square and slider.
 - Enter the desired color's hex code in the text box.

Note: To revert the color back to the default (deep pink) color, select the "here" in the tip text.

2. Select **Save**.

Fixed Dashboard Width

The default setting is **Auto** (i.e. responsive) —dashboard element arrangements shift for different sized device screens and browser windows. Setting the width to a fixed number of columns can help dashboard elements to remain in intentional arrangements.

To set a standard fixed width for all existing and new dashboards.

1. From the dropdown menu, select the desired number of columns, or enter the number.

Note: A column is the width of one medium-sized card (for example, one Weather card).

2. Select **Save**.

Note: A **Dashboard Width** set for an individual dashboard overrides the **Fixed Dashboard Width** set here. (See [Setting a Dashboard's Width on page 55](#).)

Note: Elements on an preexisting dashboard without an individually set **Dashboard Width** may shift from an intended arrangement to accommodate the new **Fixed Dashboard Width**.

Note: A left-right scroll bar will appear for dashboards on narrower screens and browser windows.

Measurements

1. From the dropdown menu, select the default unit type (**Metric**, **Imperial**, or **Mixed**) to use for displaying point values on cards, trends, etc.
2. Select **Save**.

Security

Session Inactivity Timeout

1. From the dropdown menu, select the duration of time that no activity can be detected before requiring a login again.

Note: **None** means that the session will never timeout due to inactivity.

2. Select **Save**.

Minimum password length required

1. Enter the desired minimum number of characters to require for a password.
2. Select **Save**.

Running Jobs

Running Jobs is a diagnostic tool showing a snapshot of any current processes. Most processes are completed within a few minutes. During initial discovery of a large network, processes may last considerably longer. Any job that lasts more than a few hours, however, is probably stuck.

Canceling a “stuck” or pending job (from app.kmccommander.com)

1. Select **Delete**  next to the running job.
2. In the Delete Running Job dialog, select **Reboot and Delete**.

Note: A countdown timer appears for 2 minutes and 30 seconds in an orange box at the bottom of the screen (over the Save button) while the KMC Commander gateway reboots.

Note: To access the **Save** button during the reboot process, you can close the countdown timer. The reboot process will still continue.

3. If you need to cancel more running jobs, select **Delete**  next to them.

Note: If deleted during the 2 minutes and 30 seconds that the gateway is rebooting, the jobs will delete without needing to confirm.

Gateway Information

Element	Meaning / Additional Information
Box Service Tag	Matches the service tag number found on the bottom of the gateway of the project currently accessed. It is the last seven digits, after "CommanderBX".
Last logged communication time	Shows the time of the last logged communication at the time that the web browser loaded the page.
Data usage	Shows the year and month (the last complete month) for which data usage information is displayed, as well as the amount of received data (RX) and transmitted data (TX) in gibibytes (GiB).
Reboot Gateway	Selecting Reboot Gateway initiates rebooting of the KMC Commander gateway. A timer counts down for 2 minutes and 30 seconds, during which Reboot Gateway is unavailable. Note: The gateway must have a Cloud connection to perform a remote reboot.

License Information

Element	Meaning / Additional Information
Name	The project name associated with the license in the KMC Commander license server.
Expiration Date	See "How Does Licensing Work?" in the KMC Commander (Dell or Advantech gateway) data sheet for details.
Automated Billing	Contact a KMC Controls sales representative or customer service to turn automated billing on or off . See Contact Information on page 166.

Element	Meaning / Additional Information
Licensed Points	The maximum number of points of interest that can be trended and/or written to by KMC Commander under the current license.
Used Points	The number of data points currently configured to be trended and/or written to by KMC Commander as points of interest.

System Integrator

The name of the System Integrator associated with the project in the KMC Commander license server displays here.

Enabled Addons

A list of the add-ons (extra features) purchased for this license displays here. (See [Add-ons \(and Data Explorer\) on page 141.](#))

Configuring Protocol Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Accessing Protocol Settings

Go to **Settings** , then **Protocols**.

Individual Point Intervals

The [Point Update Wait Interval \(Minutes\) on page 15](#) determines the default trending frequency for all points of interest in the project. However, project needs may require some points to trend at a lower or higher frequency. For those cases, you can configure **Low**, **Medium**, and **High** options (independent of the **Point Update Wait Interval**). When [Assigning Device Profiles on page 43](#) or [Editing a Device Profile on page 45](#), you can then select the **Low**, **Medium**, or **High** option from a **Trending Frequency** drop-down menu for the required points.

Low

Low configures the **Low** option of the **Trending Frequency** drop-down menu (found when [Assigning Device Profiles on page 43](#)).

1. Enter the longer interval (in minutes) at which some points in the project need updated (polled).

Note: The longest interval allowed is 60 minutes.

2. Select **Save**.

Medium

Medium configures the **Medium** option of the **Trending Frequency** drop-down menu (found when [Assigning Device Profiles on page 43](#)).

1. Enter the medium interval (in minutes) at which some points in the project need updated (polled).

Note: **Medium** is independent of the [Point Update Wait Interval \(Minutes\)](#) on page 15 (the default point polling interval for all points of interest in the project).

2. Select **Save**.

High

High configures the **High** option of the **Trending Frequency** drop-down menu (found when [Assigning Device Profiles](#) on page 43).

1. Enter the shorter interval (in minutes) at which some points in the project need updated (polled).

Note: The shortest interval allowed is 0.5 minutes.

2. Select **Save**.

BACnet

Device Instance

The device instance of the local KMC Commander gateway can be changed here.

Note: A manual restart is required for a change to take effect.

To change the device instance:

1. Enter a new device instance.
2. Select **Save**.

Max Invoke ID

The KMC Commander gateway uses the Max Invoke ID to send multiple requests without waiting for responses, until the Invoke ID limit (the entered value) is reached.

Note: A value of 1 means the KMC Commander gateway will always wait (or timeout) for a response before setting the next request in its queue.



Caution: The KMC Commander gateway will use multiple UDP ports for its Source Port in sending messages if greater than 1. It will always use the configured UDP port to talk to devices, but will use different UDP ports to receive the responses. These ports start with 47808 and go up consecutively. Do not set Invoke ID to anything greater than 1 if your firewall blocks these ports.

To change the Max Invoke ID (from the default of 1):

1. Enter a new value (1 to 5 maximum requests).
2. Select **Save**.

Read Priority Array Wait Interval (Minutes)

The Read Priority Wait Interval is the time between updates (polling) of priority array values.

Note: This interval affects how quickly an indication that a point is in override might display on cards. (See [Display Point Override on page 10](#) in **Settings**  > **Project**.) It also affects how up-to-date **Manual Override** reports will be. (See [Configuring a Manual Override Report on page 130](#).)

To change the Read Priority Array Wait Interval (from the default of 60 minutes):

1. Enter a new value (0 to 180 minutes).

Note: Setting to 0 will disable the priority array reading daemon (background polling process) and values will not update.

2. Select **Save**.

BACnet/Niagara

Point Update Wait Interval (Minutes)

The Point Update Wait Interval is the default time between updates (polling) of points on trends, alarms, and any reads through an API.

To change the Point Update Wait Interval (from the original default of 5 minutes):

1. Enter a new value (1 to 60 minutes).
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

Manual Write Timeout

Manual Write Timeout sets the default choice of duration for any manual overrides made of setpoints or other objects on dashboards.

Note: The default duration is **Permanent**, meaning manual overrides will continue indefinitely until the next schedule change or manual override occurs.

To set the Manual Write Timeout :

1. Select the manual override duration (15 minutes to 1 week) from the dropdown list.
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

Default Manual Write Priority

Default Manual Write Priority sets the default BACnet priority choice used to write manual changes from the dashboard.

To change the Default Manual Write Priority (from the default of 8):

1. Enter a new BACnet priority value.
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

Schedule Write Priority

Schedule Write Priority is the BACnet priority used to write normal (i.e., not holiday) schedule events.

Note: If KMC Commander schedules will be used to control devices, this value must be higher than the default schedule write priority values in the controlled devices. (See [Managing Schedules and Events on page 94.](#))

To change the Schedule Write Priority (from the default of 16):

1. Enter a new BACnet priority value.
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

Holiday Schedule Write Priority

Holiday Schedule Write Priority is the BACnet priority used to write holiday schedule events.

Note: If KMC Commander schedules will be used to control devices, this value must be higher than the default schedule write priority values in the controlled devices. (See [Managing Schedules and Events on page 94.](#))

To change the Holiday Schedule Write Priority (from the default of 15):

1. Enter a new BACnet priority value.
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

Override Schedule Write Priority

Override Schedule Write Priority is the BACnet priority used to write override schedule events.

To change the Override Schedule Write Priority (from the default of 8):

1. Enter a new BACnet priority value.
2. Select **Save**.

Note: Niagara settings can take up to 15 minutes to take effect.

KMDigital

Note: KMC Commander supports KMDigital through use of a [KMD-5551E](#) translator.

Manual Write Priority (KMD Devices)

This is the priority used to write manual changes from the dashboard to KMDigital devices through the translator.

Note: KMDigital controllers have only manual or auto write “priorities.” The translator enables a virtual priority array on the KMDigital device points by mapping them inside the translator. Auto (priority 0) is the default behavior for KMDigital, and setting any other priority will write to the KMDigital device in manual mode. See the “Translation concepts” section in the [KMD-5551E](#) translator’s application guide for more information.

To change the Manual Write Priority (from the default 0 [Auto]):

1. Enter a new priority value.
2. Select **Save**.

Schedule Write Priority (KMD Devices)

This is the priority used to write schedule events to KMDigital devices through the translator.

Note: KMDigital controllers have only manual or auto write “priorities.” The translator enables a virtual priority array on the KMDigital device points by mapping them inside the translator. Auto (priority 0) is the default behavior for KMDigital, and setting any other priority will write to the KMDigital device in manual mode. See the “Translation concepts” section in the [KMD-5551E](#) translator’s application guide for more information.

To change the Schedule Write Priority (from the default 0 [Auto]):

1. Enter a new priority value.
2. Select **Save**.

Miscellaneous

Shorten JACE Format Point Names

1. For Niagara Networks, choose whether or not to automatically shorten JACE format point names:
 - If turned **Off**, each point name read from a JACE can be extremely long and include a variety of additional device information.
 - If turned **On**, (the default) the name shortens to just the names of the points themselves (i.e. the third-to-last and last segments of the object name).
2. Select **Save**.

SNMP MIB Files

To upload a MIB file for SNMP devices:

1. Select **Upload**.

2. In the **Upload SNMP** window, select **Choose file**.
3. Locate the MIB file.
4. Select **Upload**.

Adding and Configuring Users

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Adding a User

1. Go to **Settings** , **Users/Roles/Groups**, then **Users**.
2. Select **Add New User**.
3. In the **Add New User** window, enter the First Name, Last Name, and Email address of the user.
4. Select the user's **Role** from the dropdown menu.

Note: Permissions for roles are defined in the **Roles** settings. (See [Configuring Roles on page 23](#).)

5. Enter the user's **Office Phone** and **Cell Phone**.

Note: If you want the user's cell phone to be used for SMS alarm messages, switch on **Use Cell Phone for SMS**.

6. If **Alarm Groups** have been set up, you can (optionally) assign the user to one now from the dropdown. (See [Configuring \(Alarm Notification\) Groups on page 25](#).)
7. Select **Add**.

Note: The new user appears in the list (displayed under **Users**).

Note: For information on how to add multiple user instances to multiple projects using a .xlsx (Microsoft Excel) file, see [Bulk Editing Users on page 19](#).

Configuring a User's Topology Access

Once a site topology has been set up in **Site Explorer** (see [Creating a Site Topology on page 47](#)), you can allow a user to access certain devices and not others.

Note: Access to all devices is the default.

To edit a user's topology access:

1. After [Adding a User on page 18](#), from the right end of the user's row, select **Edit Topology** .
2. In the **Edit Topology Access** window:

- To remove the user's access to devices, clear the checkbox in front of the device, zone, floor, building, or site.
- To grant the user access to devices, select the checkbox in front of the device, zone, floor, building, or site.

Note: Clearing a checkbox for a zone, floor, building, or site will automatically clear check boxes for all the devices underneath it in the topology.



Caution: Administrators who clear devices in their own profiles and save their profiles will not be able to see those devices again to restore their own access. Another administrator, however, may be able to restore the other's access. Otherwise, the device will need to be rediscovered as a new device.

3. Select **Apply** at the bottom (you may need to scroll down to see it).

Editing Users

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Editing a User

1. Go to **Settings**  > **Users/Roles/Groups** > **Users**.
2. In the row of the user that you want to edit, select **Edit User** .
3. In the **Edit User** window, modify the user configuration as needed. (See [Adding and Configuring Users on page 18](#) for more information).
4. Select **Save**.

Bulk Editing Users

You can bulk edit multiple user instances for multiple projects by uploading a .xlsx (Microsoft Excel) file. The feature helps you manage *all* users for *all* projects under the control of your System Integrator account.

To avoid confusion and throwing errors (see [Error Messages on page 23](#)) we recommend that you:

- Download a fresh, current template immediately prior to bulk editing users. (See [Download and open the template on page 20](#).)
- Do not allow other users on your team to upload your template file—have them download their own template file.

Access the Bulk User window

1. Go to **Settings**  > **Users/Roles/Groups** > **Users**.
2. Select **Bulk User Edit**, which opens the **Bulk User** window.

Note: Though you access the **Bulk User** window from within a single project, the feature helps manage *all* users for *all* projects under the control of your System Integrator account.

Download and open the template

1. Select **Download Template with Current Users**.

Note: This causes the template file—*bulk-user-edit-template.xlsx*—to generate. The template contains the configurations of all users for all projects under the control of your System Integrator account (at that moment).

2. Locate and open the template file.

Note: The template file—*bulk-user-edit-template.xlsx*—downloads to the place your browser designates for file downloads.

3. Enable editing of the template file.

Continue by [Adding User Instances on page 20](#), [Deleting User Instances on page 21](#), and/or [Changing Users' Roles on page 22](#).

Adding User Instances

1. In a new row of the spreadsheet, fill out the columns:

Column Label	Explanation	Required?
firstName	Enter the first name of the user that you want to add.	Yes
lastName	Enter the last name of the user that you want to add.	Yes
email	Enter the email address of the user.	Yes
role	Enter the role that you want the user to have. (See Configuring Roles on page 23 for more information.)	Yes

Column Label	Explanation	Required?
projectId	Enter the identification code of a project that you want to add the user to. (You can copy the projectId from another user row where it is already associated with the projectName that you know.) If you want to add the user to multiple projects, fill out multiple rows—one for each project. Note: The projectId is the unique identifier to ensure that the system finds the exact project.	Yes
projectName	You can copy the projectName from another user row for consistency. However, if you upload the .xlsx file with the projectName blank, the system will automatically fill in the projectName associated with the projectId. (If you then Download and open the template on page 20 again, you will see the projectName filled in.) Note: If you enter the projectName but leave the projectId blank, the user cannot be added. (The projectId is the unique identifier to ensure that the system finds the exact project.)	No
delete	Enter FALSE, or leave blank.	No
sendNotificationEmail	The user will receive an invitation or notification email if you enter TRUE.	No

- Repeat step 1 for as many user instances as you want to add in one bulk user edit.

When you are done modifying the spreadsheet, [Save and upload the file on page 22](#).

Deleting User Instances

- In the row of each user instance that you want to delete, enter TRUE in the **delete** column.

Note: If you want to completely remove a user from KMC Commander, enter TRUE in the **delete** column for every instance of that user associated with any project.

2. If you want a user to receive an email notifying them that they were removed from a project, enter TRUE for **sendNotificationEmail**.

When you are done modifying the spreadsheet, [Save and upload the file on page 22](#).

Changing Users' Roles

1. For each user instance that you want to change, enter an alternative, valid role in the **role** column. (See [Configuring Roles on page 23](#) for more information.)
2. If you want a user to receive an email notifying them that their role was updated for that project, enter TRUE for **sendNotificationEmail**.

When you are done modifying the spreadsheet, [Save and upload the file on page 22](#).

Save and upload the file

1. Save the .xlsx file.

Note: You can save the file with a new name; the system will still accept it.

2. In the **Bulk User** window of KMC Commander, select **Choose file**.
3. Locate and select the saved file.
4. Choose whether or not the system should **Stop process on errors**.

Note: If **Stop process on errors** is checked, the system will not process any rows after an error occurs.

5. Select **Upload**.

Note: This causes an output file—*output.xlsx*—to generate. It downloads to the place your browser designates for file downloads.

6. Check the output file for [Success Messages on page 22](#) and [Error Messages on page 23](#).

Success Messages

successMessage	Explanation
User invited successfully	You invited an entirely new user to KMC Commander with this project.
User added successfully	You invited an existing user (of at least one project) to another project.

successMessage	Explanation
User removed successfully	You removed a user from a project. (To completely remove a user from KMC Commander, repeat for all of their projects.)
User is already removed from the project	You tried to remove a user instance that was already removed. (Relax.)
User role updated successfully	You updated a user's role for a project.
Duplicated row, no action taken	You accidentally made two identical rows in the file. The action was taken the first time. (Relax.)

Error Messages

errorMessage	Explanation / Remedy
Missing required fields	Fill out (at minimum) firstName , lastName , email , role , and projectId .
Project not found	Enter a valid projectId . Copy and paste the needed projectId from an existing row.
User does not have access to project	The "user" in this case is you. You do not have access to the project associated with the projectId that you entered. Or you have access, but are assigned a role without Admin permissions. Obtain access (with Admin permissions) from an Admin of that project.
User does not exist	You tried to delete a user who does not exist in the system (relax). If you intended to add the user, enter FALSE for delete .
Role does not exist	Enter a role that is configured for the project. (See Configuring Roles on page 23 .)

Configuring Roles

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Adding a New Role

KMC Commander comes with four preset roles (**Admin**, **Owner**, **Technician**, and **Occupant**). In addition, you can create custom roles.

To create a new custom role:

1. Go to **Settings** , **Users/Roles/Groups**, then **Roles**.
2. Select **Add New Role**.
3. Enter a name for the new role.
4. Select **Add**.
5. Define that role by choosing the features you want to give that role access to. (See [Defining Roles on page 24.](#))
6. Select **Save**.

Defining Roles

1. Go to **Settings** , **Users/Roles/Groups**, then **Roles**.
2. Choose the KMC Commander features that you want to give a role access to (see the table below) by checking the boxes for those features in the row for that role.
3. Select **Save**.

Note: To apply a role to a user, see [Adding and Configuring Users on page 18.](#)

Note: The **Admin** role is permanently set to have **Admin permissions**, giving those users access to all features (including **Settings** ).

Note: See [Configuring a User's Topology Access on page 18](#) for information on that separate process.

Column Label	What it Does
Admin	If <i>Admin permissions</i> are selected for a role, those users will have full access to all features (including Settings ) , whether the other features' checkboxes are selected or not.
Dashboard	Selecting this for a role gives those users access to Dashboards  (which displays cards and decks). Clearing this hides Dashboards  from their side navigation menu. (See Dashboards and Their Elements on page 54.)

Column Label	What it Does
Networks	Selecting this for a role gives those users access to Networks  . Clearing this hides Networks  from their side navigation menu. (See Configuring Networks on page 37.)
Schedules	Selecting this for a role gives those users access to Schedules  . Clearing this hides Schedules  from their side navigation menu. (See Managing Schedules and Events on page 94.)
Alarms	Selecting this for a role gives those users access to Alarms  . Clearing this hides Alarms  from their side navigation menu. (See Managing Alarms on page 112.)
Trends	Selecting this for a role gives those users access to the Trends  setup. Clearing this hides Trends  from their side navigation menu. (They can still view trend cards on a dashboard.) (See Managing Trends on page 102.)
Data Explorer	Selecting this for a role gives those users access to Data Explorer . Clearing this hides Data Explorer from their side navigation menu (in Add-ons ). (See Using Data Explorer on page 141.)
Hide Card Detail	If selected for a role, those users will not be able to flip over dashboard cards.
Read Only	If selected for a role, those users will only be able to view (not edit) dashboards.
Dashboard Autoshare	The dashboards of the user that you select from the dropdown list (the source user) will be autoshared (copied) as templates with any new users given this role. When new users with this role are added to the project, their dashboards will populate with the templates (as they are at that moment). Subsequent changes by the source user to the dashboards will not reflect in the accounts of the users that they were autoshared with. Likewise, the new users can modify the populated dashboards without affecting the source user's templates. It is recommended to make template accounts to serve as the source "user", rather than using an individual's account.

Configuring (Alarm Notification) Groups

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Adding a Group Name

1. Go to **Settings** , **Users/Roles/Groups**, then **Groups**.
2. Select **Add New Group**.
3. Enter a name for the group.
4. Select the **Add New Group**.

Note: When you are done adding new group names, you can close the tool from the far right of the row.

5. Continue by [Adding Users to a Group on page 26](#).

Adding Users to a Group

1. After [Adding a Group Name on page 26](#), select **Edit**  in the group's row.
2. In the **Edit [Group Name]** window, select the checkboxes next to the users you want to include in the group.

Note: You can sort the list of names by choosing an option (**Email domain**, **Email**, **First Name**, **Last Name**, or **Role**) from the **Sort By** dropdown menu. You can also narrow the list by entering a name, email, or role in the search field.

3. Select **Save**.

For a user to receive an alarm notification, their **Notification Group** must be selected when [Configuring a Point Value Alarm on page 112](#).

Configuring Weather Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Accessing Weather Settings

Go to **Settings** , then **Weather**.

Temperature

Select **Fahrenheit** or **Celsius** to set the temperature unit type that will display on weather cards.

Weather stations

For Weather cards on the Dashboard, you must first add weather stations to this list. The listed weather stations will appear in a dropdown list on Weather cards.

To add a new station:

1. Select **Add New Station**.

2. Choose whether to search by **City** or **ZIP** code.

Note: If searching by **City**, make sure the country the city is located in is selected from the dropdown menu (US = United States; AU = Australia; CA = Canada; GB = Great Britain; MX = Mexico; TR = Turkey)

3. Enter the city's name or the ZIP code.
4. Choose the desired city from the list that appears.
5. Select **Add**.

Searching User Action Logs

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

User action logs allow viewing of when modifications were made by a user (or by API calls) to networks, profiles, devices, schedules, and writable points.

Accessing User Action Logs

Go to **Settings** , then **User Action Logs**.

Finding User Actions

The most recent changes are at the top of the list. Use the forward arrow at the bottom to see older action log pages.

Note: In the **Object (Name)** column, the first word is the Object Type (e.g., network, point, schedule) and the text inside the parentheses is the Object Name.

To narrow the list by a user's first or last name:

1. Enter the **User First Name** and/or **User Last Name**.
2. Select **Apply**.

To narrow the list by a date range:

1. Select the **Time Range** field.
2. Choose an earliest date.
3. Choose a latest date.
4. Select **Ok**.

Note: Selecting **Clear** clears the date range.

5. Select **Apply**.

To apply a filter to the list:

1. Select **Select Filters**.
2. Enter descriptions in the desired fields (for example, point (), device (), network (), schedule (), or profile () in the Object field).
3. Select the checkbox next to the description.
4. Select **Apply**.

Configuring LAN/Ethernet Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

Network Interface Port Labeling

The network interface ports are labeled differently depending on the model of KMC Commander gateway:

Dell Edge Gateway 3002	Ethernet 1 [eth0]	Ethernet 2 [eth1]	Wi-Fi [wlan0]
Advantech UNO-420	LAN B [enp1s0] (PoE In)	LAN A [enps2s0]	Wi-Fi [wlp3s0]

Configuring the LAN/Ethernet Settings

Only one LAN/Ethernet port should have a live *Internet* connection. The ports should not have the same IP addresses.

1. Go to **Settings** , **Network Interfaces**, then **LAN B [enp1s0] (PoE In)** / **Ethernet 1 [eth0]**, or **LAN A [enp2s0] / Ethernet 2 [eth1]**.
2. Switch **Disabled** to **Enabled** (if not already).
3. Enter the information in the boxes below as needed.
4. Choose the **Network Area Type** (**LAN** or **WAN**).
5. If the gateway will primarily access the cloud through a cellular connection and you are configuring this Ethernet port for connection to a local subnet, choose **yes** for either **Isolate IPv4 to Local Subnet** or **Isolate IPv6 to Local Subnet**.



Caution: If your local connection is routed and you select **yes**, it may disable your ability to connect to the gateway locally.

6. Select **Save**.

Configuring Wi-Fi Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

Know Before Beginning

Wi-Fi Uses

Wi-Fi is usually used as an access point only for installation, then is turned off. See [Turning off Wi-Fi \(after installation\) on page 29](#).

Wi-Fi may continue to be used as an access point. However, in that case the password should be changed from the factory default. See [Changing the passphrase \(password\) to continue using Wi-Fi as an access point on page 29](#).

Wi-Fi can also be used as a client after installation to connect to an existing Wi-Fi network. See [Using Wi-Fi \(as a client\) to connect to an existing Wi-Fi network on page 30](#).

Network Interface Port Labeling

The network interface ports are labeled differently depending on the model of KMC Commander gateway:

Dell Edge Gateway 3002	Ethernet 1 [eth0]	Ethernet 2 [eth1]	Wi-Fi [wlan0]
Advantech UNO-420	LAN B [enp1s0] (PoE In)	LAN A [enps2s0]	Wi-Fi [wlp3s0]

Turning off Wi-Fi (after installation)

1. Go to **Settings** , **Network Interfaces**, then **Wi-Fi [wlp3s0]** / **Wi-Fi [wlan0]**.
2. Switch **Enabled** to **Disabled**.
3. Select **Save**.

Changing the passphrase (password) to continue using Wi-Fi as an access point

1. Go to **Settings** , **Network Interfaces**, then **Wi-Fi [wlp3s0]** / **Wi-Fi [wlan0]**.
2. Leave the switch on **Enabled**.
3. Leave **Access Point** selected for **AP Mode**.
4. Edit the Wi-Fi information as needed.

Note: KMC Commander has a built-in DHCP server. Using **DHCP Range Start** and **DHCP Range End**, set the range of available addresses for devices to connect to the access point.

5. Change the default **Passphrase** (a.k.a. password).

Note: The new password should have at least eight characters, be mixed case, and use at least one number.

6. Record the new password and any new addresses.
7. Switch **Internet Sharing** to either **Enabled** or **Disabled**.

Note: If **Enabled**, devices connected to the KMC Commander gateway by this wireless access point can access the Internet through the gateway, in addition to accessing the KMC Commander user interface.

Note: If **Disabled**, devices connected to the KMC Commander gateway by this wireless access point will be able to access the KMC Commander user interface only.

8. Select **Save**.

Using Wi-Fi (as a client) to connect to an existing Wi-Fi network

1. Go to **Settings** , **Network Interfaces**, then **Wi-Fi [wlp3s0]** / **Wi-Fi [wlan0]**.
2. Switch **Enabled** to **Disabled**.
3. Select **Save**.
4. Restart the gateway. (See [Restarting the Gateway on page 162](#).)
5. Return to **Wi-Fi [wlp3s0]** / **Wi-Fi [wlan0]**.
6. Switch **Disabled** back to **Enabled**.
7. For **AP Mode**, choose **Client**.
8. For **Type**, choose **DHCP** or **Static** as needed.
9. Edit the Wi-Fi information as needed.
10. Select **Save**.

Note: While in Client mode, selecting **Show available networks** shows information about all Wi-Fi signals that the KMC Commander gateway is receiving.

Configuring Cellular Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure Cellular Settings

Note: Cellular settings are only available on KMC Commander Dell cellular model gateways supplied with a SIM card.

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

Only one port (Ethernet or cellular, but not both) should have a live Internet connection.

1. Activate the supplied SIM card and install the cellular antennas if this has not already been done.

Note: See "Installing Optional Cellular and Memory" in the *KMC Commander Dell Gateway Installation Guide*.

2. Go to **Settings** , **Network Interfaces**, then **Cellular [cdc-wdm0]**.
3. Switch **Disabled** to **Enabled** (if not already).
4. Enter the **Access Point Name** (APN) supplied by the cellular carrier.

Note: Usually the APN will be "vzwinternet" for Verizon or "broadband" for AT&T. For a Verizon static IP, it will be a variation of "xxx.vzwstatic" dependent on location.

Note: Leave the Route Metric (Priority) at its default.

5. Select **Save**.

Note: When a cellular connection is made, an IP address appears.

Configuring Date and Time Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

During installation, if the network does not provide an initial NTP time service, a different time server can be entered here to allow initial setup of the system.

Selecting a Time Zone

1. Go to **Settings** , **Network Interfaces**, then **Date & Time**.
2. Switch **Disabled** to **Enabled** (if not already).
3. From the **Time Zone** dropdown list, choose a time zone. (See [About UTC Time Zones on page 32](#).)

Note: To narrow the list of time zones, clear the text in the dropdown list selector, then enter a geographical area.

4. Select **Save**.

Note: The project time zone can also be set under **Projects** in KMC Commander System Administration. See [Accessing System Administration on page 5](#).

Entering an NTP (Network Time Protocol) Server

Note: An NTP server provides accurate, synchronized time.

1. Go to **Settings** , **Network Interfaces**, then **Date & Time**.
2. For **NTP Server**, enter the server's address.

Note: Leave the **NTP Fallback Server** default address (ntp.ubuntu.com) unless a definite alternative is known.

3. Select **Save**.

About UTC Time Zones

UTC (Coordinated Universal Time) is also known as *GMT (Greenwich Mean Time)*, *Zulu*, or *Z time*. KMC Commander may display the date (for example, 2017-10-11) and the time in 24-hour UTC format (for example, T18:46:59.638Z, which means 18 hours, 46 minutes, and 59.638 seconds in the Coordinated Universal Time zone). UTC is, for example, 5 hours ahead of Eastern Standard Time or 4 hours ahead of Eastern Daylight time.

See the table below for more time zone conversions:

Sample Time Zones*	Offset from UTC (Coordinated Universal Time) to Equal Local Time**
American Samoa, Midway Atoll	UTC−11hours
Hawaii, Aleutian Islands	UTC−10hours
Alaska, French Polynesia	UTC−9 hours (or 8 hours with DST)
USA/Canada Pacific Standard Time	UTC−8 hours (or 7 hours with DST)
USA/Canada Mountain Standard Time	UTC−7 hours (or 6 hours with DST)
USA/Canada Central Standard Time	UTC−6 hours (or 5 hours with DST)
USA/Canada Eastern Standard Time	UTC−5 hours (or 4 hours with DST)

Sample Time Zones*	Offset from UTC (Coordinated Universal Time) to Equal Local Time**
Bolivia, Chile	UTC−4 hours
Argentina, Uruguay	UTC−3 hours
United Kingdom, Iceland, Portugal	0 hours
Europe (most countries)	UTC +1 hour
Egypt, Israel, Turkey	UTC +2 hours
Kuwait, Saudi Arabia	UTC +3 hours
United Arab Emirates	UTC +4 hours
Maldives, Pakistan	UTC +5 hours
India, Sri Lanka	UTC +5.5 hours
Bangladesh, Bhutan	UTC +6 hours
Laos, Thailand, Vietnam	UTC +7 hours
China, Mongolia, Western Australia	UTC +8 hours
Korea, Japan	UTC +9 hours
Central Australia	UTC +9.5 hours
Eastern Australia, Tasmania	UTC +10 hours
Vanuatu, Solomon Islands	UTC +11 hours
New Zealand, Fiji	UTC +12 hours
*Minor parts of the named areas may be in other time zones. **May also need to convert from 24 to 12 hour format. Zulu or Greenwich Mean Time is the same as UTC for practical applications.	

Configuring Whitelist/Blacklist Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

Know Before Beginning



Caution: Deleting any of the default listings is not recommended. Deleting the wrong listing could result in loss of communication with the gateway.

For both Ethernet ports, the default setting for Whitelist/Blacklist **Network Area Type** is LAN. A LAN (Local Area Network) is generally not publicly accessible on the Internet. A WAN (Wide Area Network) generally is.

The *whitelist* contains addresses that are *always* allowed inbound access, and the *blacklist* contains addresses that are *never* allowed inbound access. The whitelist and blacklist apply only to unsolicited *inbound* requests. *Outbound* messages have no blocks.

Addresses and *ports* can be added to the whitelist. For BACnet, the UDP port for traffic may need to be added to the UDP Port (Whitelist) section if it is not already in the list.

For remote access into a gateway via VPN, the VPN subnet might need to be added to the LAN whitelist. Add a subnet as a range of addresses, not a single address.

For IP addresses, enter an address or a range, with the range defined with the subnet mask length using CIDR (Classless Inter-Domain Routing) notation. (For example, enter the base address, followed by a slash, and then the subnet mask length as the number of most significant bits of the IP address, such as 192.168.0.0/16.)

Adding an IP Address to a Whitelist or Blacklist

1. Go to **Settings** , then **Whitelist/Blacklist**.
2. Select the **IP Address** box that is below **Whitelist IP** or **Blacklist IP** for the network type (**LAN** or **WAN**) that you want to add the address to.
3. Enter the IP address.

Note: To enter a range of IP addresses, define the range with the subnet mask length using CIDR notation. (For example, enter the base address, followed by a slash, and then the subnet mask length as the number of most significant bits of the IP address, such as 192.168.0.0/16.)

4. Select **Add**.
5. Select **Save**.

Entering Allowed TCP and UDP Ports

1. Go to **Settings** , then **Whitelist/Blacklist**.
2. Select the textbox below either **TCP Port (allow)** or **UDP Port (allow)**.
3. Enter the port number(s).

Note: Separate port numbers with a comma (,). For example: 53,67,68,137.

Note: Use a colon (:) to enter a range of ports. For example, 47814:47819.

4. Select **Save**.

Configuring IP Tables

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

The **IP Tables** list is a master override whitelist of the LAN/WAN lists for Cloud connectivity.



Caution: Deleting any of the default listings is not recommended. Deleting the wrong listing could result in loss of communication with the gateway.

Adding to IP Tables

1. Go to **Settings** , then **IP Tables**.
2. In **IP Address**, **TCP Ports**, and/or **UDP Ports**, enter the relevant IP address and connected ports as needed.

Note: Enter an address or a range with the range defined with the subnet mask length using **CIDR** (Classless Inter-Domain Routing) notation. (For example, enter the base address, followed by a slash, and then the subnet mask length as the number of most significant bits of the IP address, such as 192.168.0.0/16.)

3. Select **Save**.

Configuring Proxy Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only configure these settings when logged into the gateway *locally*. See [Logging In at the Job Site](#).

If required for this KMC Commander gateway:

1. Go to **Settings** , then **Proxy**.
2. Enter the **HTTP Proxy Address** and the **HTTPS Proxy Address**.
3. Select **Save**.

Configuring SSH Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For increased security, you can only enable SSH when logged into the gateway *locally*. See [Logging In at the Job Site](#).

KMC Commander's remote SSH (Secure Shell) login access is primarily for technical support representatives using a terminal emulator to provide troubleshooting or system configuration. For security, remote terminal access is disabled by default.

Only when remote terminal access is needed:

1. Go to **Settings** , then **SSH**.
2. Switch **Disabled** to **Enabled**.

Configuring Networks

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Supported Network Protocols

KMC Commander can connect to these protocols:

- BACnet IP (directly)
- BACnet Ethernet (directly)
- BACnet MS/TP (with a [BAC-5051AE](#) BACnet Router)
- KMDigital (with a [KMD-5551E](#) Translator or a KMDigital controller with a BACnet Ethernet interface)
- Modbus TCP (directly, with an imported Modbus register map CSV file)
- SNMP (directly, with an imported MIB file)
- Node-RED (with an additional license, installation of Node-RED, and custom programming).

Configuring a BACnet Network

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Before Configuring a BACnet *MS/TP* Network

BACnet devices on an *MS/TP* network require a [BAC-5051AE](#) BACnet router for (IP or Ethernet) connection to the KMC Commander IoT gateway. See [BAC-5051AE](#) instructions for connecting MS/TP devices to a KMC Commander network.

Note: The KMC Commander IoT gateway is not a BACnet router or a BACnet device. (Nevertheless, a 4194303 Device ID with “SimpleClient” may appear in the Network Manager of KMC Connect or TotalControl.)

Configuring a BACnet Network

1. Go to **Networks Explorer** , then **Networks**.
2. Select **Configure New Network** to go to the **Configure Network** page.
3. For **Protocol**, select **BACnet**.
4. For **Data Layer**, choose **IP** or **Ethernet**.
5. Enter the network name and address information.

Note: Network information is dependent on the site survey and the building's IT.

Note: Be sure that port and network numbers are correct. Multiple networks may be required to see all devices. If BACnet devices are on the local network, do not enter the router's IP address.

6. Optionally, choose **Single** or **Range** for **Instance Filter Option**.

Note: Entering a known range of device instances will speed up the later discovery process. If devices are not found as expected, try expanding the range or choose **Any**.

7. Select **Save**.

Continue by [Configuring Devices on page 43](#).

Configuring a KMDigital Network

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Know Before Beginning

KMC Commander can discover points within KMDigital controllers (depending on the controller models and network configurations):

- Using Tier 1 KMDigital controllers with BACnet Ethernet interfaces. (Only Tier 1 points are available—not points of connected Tier 2 controllers. No KMD-5551E Translator or Niagara network is required.)
- Using an existing KMC KMD-5551E Translator on a properly licensed Niagara network. (Tier 1 and 2 points are available.)
- Using a KMD-5551E Translator and a Translator license for KMC Commander. (Tier 1 and 2 points are available. No Niagara network is required.)

Note: Only KMDigital points and their values are available through the KMD-5551E Translator. KMDigital trends, alarms, and schedules are not available.

Note: See the [KMD-5551E Translator](#) documentation for instructions on how to install and use it on a KMDigital network.

Four Tier 1 KMDigital controller models have BACnet Ethernet interfaces. Their points are discoverable in KMC Commander as *virtual* BACnet objects using BACnet *Ethernet* protocol (without a KMD-5551E Translator or Niagara). (Points in any Tier 2 controllers connected to them by EIA-485 wiring, however, are not discoverable without a KMD-5551E)

The Tier 1 models with BACnet interfaces are:

- KMD-5270-001 WebLite Controller (discontinued)
- KMD-5210-001 LAN Controller (discontinued)
- KMD-5205-006 LanLite Controller (discontinued)
- [KMD-5290E LAN Controller](#)

Other KMC KMDigital devices can be discovered as virtual BACnet devices using a KMD-5551E Translator.

Through an existing KMD-5551E Translator on a properly licensed Niagara network, points on KMDigital (Tier 1 and 2) controllers will appear as virtual BACnet objects. They are discoverable like regular BACnet objects. See [Configuring a BACnet Network on page 37](#).

Without Niagara, a license to use the KMD-5551E with KMC Commander must be purchased from KMC Controls. (A KMD-5551E license for Niagara will not work as a license for the KMC Commander IoT gateway.)

Discovering KMDigital devices through a KMD-5551E without Niagara

1. Go to **Networks Explorer** , then **Networks**.
2. Select **Configure New Network** to go to the **Configure Network** page.
3. For **Protocol**, select **BACnet**.
4. For **Data Layer**, choose **IP** or **Ethernet** as needed (see above).
5. Enter the network **Name** and address information.

Note: Network information is dependent on the site survey and the building's IT.

6. Optionally, choose **Single** or **Range** for **Instance Filter Option**.

Note: Entering a known range of device instances will speed up the later discovery process. If devices are not found as expected, try expanding the range or choose **Any**.

7. Select **Save**.
8. Complete—for the KMD-5551E—the processes described in [Discovering Devices on page 43](#) and [Assigning Device Profiles on page 43](#).



Caution: If you don't complete those processes, the KMD-5551E translator will not receive the license refresh message sent from customer service. It will become unlicensed and will stop translating an hour after starting up.

Continue with [Configuring Devices on page 43](#).

Note: Tier 1 KMDigital controller models with BACnet Ethernet interfaces have points discoverable as virtual BACnet objects using BACnet Ethernet protocol (without a KMD-5551E Translator or Niagara), but they do not fully support BACnet priority arrays. (The priority array does not display properly with these devices.) On a dashboard, clearing a selected priority 1–8 value now relinquishes to the previous scheduled (highest level priority 0 or 9–16) value that was last written.

Note: In those three Tier 1 KMDigital controller models (see above), any value written at a priority 0 or 9–16 is assumed to be a scheduled write and stored locally. Any value written at a priority 1–8 is assumed to be a manual write (which sets the manual flag on these devices). When relinquishing a priority 1–8 (by selecting **Clear Selected** under **Show Advanced**), the last scheduled write value is written and the manual flag is removed.

Note: The KMD-5551E KMDigital to BACnet Translator fully supports priority arrays in Tier 1 and Tier 2 devices.

Configuring a Modbus Network

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure

Unlike BACnet, only one Modbus TCP **device** is added to the “**network**” during discovery according to the device information entered. For multiple Modbus devices, create multiple Modbus “networks.”

1. Go to **Networks Explorer** , then **Networks**.
2. Select **Configure New Network** to go to the **Configure Network** page.
3. For **Protocol**, select **Modbus**.
4. Enter the relevant network information in the fields.
5. **Upload** the Modbus register map CSV file for the particular Modbus TCP device:
 - A. Next to **Map File**, select **Upload**.
 - B. Select **Choose file**.
 - C. Locate the map file on your computer.
 - D. Select **Upload**.

Note: For full instructions about the Modbus TCP device options as well as sample register map CSV files, see the *Modbus Devices on KMC Commander Application Guide* (see [Accessing Other Documents on page 164](#)).

6. Choose the **Network Interface** from the dropdown list.
7. Select **Save**.

Continue with [Configuring Devices on page 43](#).

Configuring an SNMP Network

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About SNMP "Networks"

In an SNMP network, KMC Commander acts as an SNMP manager, gathering data points from agents (software modules inside devices such as routers, data servers, workstations, printers, and other IT devices) and triggering actions.

Note: Unlike BACnet, only one SNMP *device* is added to the “*network*” during discovery according to the information entered. For multiple SNMP devices, create multiple SNMP “networks.” For example, if the devices are all the same (e.g., four routers of the same model), the MIB file would be the same, but the IP address would be different for each one and would require four different “networks.”

Configuring

1. In **Settings**  > **Protocols**, upload the manufacturer's **MIB file** for the desired device. (See [SNMP MIB Files on page 17](#) in [Configuring Protocol Settings on page 13](#).)

Note: MIB (Management Information [data]Base) files contain data points describing the parameters of a particular device. The MIB file should be provided by the device manufacturer, and the file is uploaded to the manager (KMC Commander) so that the manager can decipher received data from the device.

2. Go to **Networks Explorer** , then **Networks**.
3. Select **Configure New Network** to go to the **Configure Network** page.
4. For **Protocol**, select **SNMP**.
5. Choose the SNMP **Protocol Version** used:
 - **v1** (simplest, oldest, and least secure).
 - **v2c** (has additional features and the largest installed base)
 - **v3** (most secure, the current standard, and recommended for use whenever possible)
6. Enter the network's **Name**.
7. Enter the **Device IP Address**.
8. Optionally, enter any **Subtree(s)**.
9. Enter the number for the **Destination Port** and **Trap** (notifications) **Port** if needed. (See the device's instructions.)

Note: The **Destination Port** (default 161) is the port in the SNMP agent (the device) that receives requests from the manager. The **Trap Port** (default 162) is the port in the manager (KMC Commander) that receives unsolicited notifications from the agents.

10. Select and enter user and security information as needed.

Note: Security settings are typically found in the SNMP device's documentation or web management page. Use the highest security supported by the device (Auth Priv being the highest, with required authentication of users and encryption of messages). If the device documentation specifies only one read or one write password but supports v3 Auth Priv, try using the same password for both Auth and Privacy fields. If there is trouble connecting to a v3 device, and the documentation does not specify an Auth or Priv protocol, try switching one or both of those protocols.

11. Select **Save**.
12. Continue with [Configuring Devices on page 43](#).

Configuring a Node-RED Network

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Node-RED "Networks"

Node-RED supports specific IP devices with programs developed by KMC Controls.

Note: Unlike BACnet, only one **device** is added to a Node-RED “**network**” during discovery, according to the device information entered. For multiple devices, create multiple Node-RED “networks”.

Before Configuring

Using Node-RED for discovery of devices requires installation of Node-RED, an additional license, and custom programming.

Note: Configuration can also be done through a licensed Node-RED add-on. See the *KMC Commander Node-RED Application Guide* (see [Accessing Other Documents on page 164](#)).

Configuring

1. Go to **Networks Explorer** , then **Networks**.
2. Select **Configure New Network**.
3. From the **Protocol** drop-down menu, select **Node-Red**.
4. Enter the device name and address information.
5. Enter the device **Password**.
6. Choose the **Device Protocol** (**Shelly** or **WiFi_RIB**) from the dropdown list.

Note: Leaving **Default** selected does nothing.

7. If you are configuring a relay that is bound to a Binary Input, select **Relay Bound to BI**.
8.

Note: For **Shelly** device protocol, **Relay Bound to BI** is always selected by default, because Shelly devices are always bound to a Binary Input.
9. Select **Save**.
10. Continue with [Configuring Devices on page 43](#).

Configuring Devices

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Discovering Devices

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

While devices can be discovered remotely from the Cloud, being on site is useful for troubleshooting.

To discover devices, after [Configuring Networks on page 37](#):

1. Select **Discover**.
2. Optionally, in **Confirm Discover Options**, change the **Instance Min** and **Instance Max**.

Note: Narrowing device discovery to a range of known device instances speeds up the discovery process.

3. Select **Discover**.

Note: For each device KMC Commander discovers, a row will appear with the device's **Instance ID**.

Note: Select anywhere in a device row's area to expand it to see more basic information about the device.

4. Select **Get Device Details** in a device's row to get the remaining information about the device.

Note: Alternatively, select **Get All Device Details** to get the details for all discovered devices.

Continue by [Assigning Device Profiles on page 43](#) to each device that is to be included in the KMC Commander installation.

Assigning Device Profiles

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

This topic describes the process for initially assigning device profiles immediately after [Discovering Devices on page 43](#). For guidance on later changing a device's profile, see [Editing a Device Profile on page 45](#).

Each device to be included in the KMC Commander installation must have a profile. Not all discovered devices need to be included, however. Assign profiles only for devices of interest. Points of interest count as points used out of the number licensed for the project. However, trends on the points of interest do not count toward the license limit.

Note: The total number of **Points Used** out of the number licensed for the project is shown in the upper-right corner of **Networks Explorer**.

While device profiles can be assigned remotely from the Cloud, being on site can be useful for troubleshooting.

Accessing the Assign profile Page

After [Discovering Devices on page 43](#):

1. Select **Save Device** in the row of a device of interest.

Note: You must select **Get Device Details** or **Get All Device Details** first to see **Save Device**. (See [Discovering Devices on page 43](#).)

2. Select **Assign Profile** to go to the **Assign profile to [device name]** page.

If a profile *with all points properly configured* for a device already exists in the project, continue to [Assigning an Existing Device Profile on page 45](#).

Otherwise, continue to [Creating and Assigning a New Device Profile on page 44](#) or [Assigning a Device Profile Based on an Existing Profile on page 45](#).

Creating and Assigning a New Device Profile

1. From the **Assign profile to [device name]** page, select **Create New**.
2. Enter a **Name** for the device profile.
3. Select the **Device Type** from the drop-down menu.
4. From the **Point Naming** drop-down menu, choose either **Protocol Default** or **Description**.

Note: This choice affects what will appear in the **Name** column when the device's points are discovered. This is primarily for KMDigital via BACnet Ethernet applications (see [Configuring a KMDigital Network on page 38](#)). If **Description** is selected during point discovery, the point name shown on dashboard cards will be the (KMDigital via BACnet Ethernet) controller point's Description (for example, MTG ROOM TEMP) rather than the generic name (for example, AI4).

5. Select **Discover**.
6. For each point that you will track, trend, schedule and/or alarm:
 - a. Select **Select Type** to open the **Select Point Type** window.

Note: Choosing the type applies the correct Haystack tags to the point and enables its use with cards, schedules, and alarms. It also automatically selects the checkbox in the **Points of Interest** column. To search for tags after configuration, see [Using Data Explorer on page 141](#).

Note: The total number of **Points Used** out of the number licensed for the project is shown in the upper-right corner of **Networks Explorer**.

- b. Find and choose the point type using the dropdown menu, search, or tree selector.
7. For any points to be trended, also select their checkboxes in the **Trend (his)** column.

8. Optionally, select an individualized trending frequency for some points from the **Trending Frequency** drop-down menu.

Note: The values for the **Low**, **Medium**, and **High** options are configured in **Settings**  **> Protocols > Individual Point Intervals**. See the topic on [Individual Point Intervals on page 13](#) for more information.

9. After all points of interest are configured, select **Save & Assign Profile**.

Assigning an Existing Device Profile



Caution: For multiple devices using the same profile, after saving one device, wait at least three minutes before saving the profile for the next device. (This ensures that all necessary writes are made and ensures the reliability of the data and the profile.)

1. From the **Assign profile to [device name]** page, select **Select Existing Profile**.
2. Choose which **Profiles to show**: **Global only**, or **Project Only**.
3. Choose the profile from the dropdown list.
4. Select **Assign Profile**.

Assigning a Device Profile Based on an Existing Profile

1. From the **Assign profile to [device name]** page, select **Select Existing Profile**.
2. Choose which **Profiles to show**: **Global only**, or **Project Only**.
3. Choose the existing profile you want to use as the basis for a new profile from the dropdown list.
4. Make any necessary changes to the profile.
5. Select **Save Copy & Assign**.
6. Enter a name for the new profile.
7. Select **Assign & Save**.

Editing a Device Profile

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Edit a Device Profile

See also information on the related but separate process, [Editing Device Details on page 46](#).

1. Go to **Networks Explorer** , then **Networks**.
2. Select **View**  (in the row of the network that has the device with the profile that you want to edit).
3. Select **Edit Profile** (in the row of the device with the profile that you want to edit).
4. Take any of the following actions to edit the profile:

- Edit the **Name**.
- Change the **Device Type**.
- Add points of interest:
 - a. Select **Select Type** (in the row of a point you want to add), which opens the **Select Point Type** window.
 - b. Find and choose the point type using the dropdown menu, search, or tree selector.

Note: Choosing the type applies the correct Haystack tags to the point and enables its use with cards, schedules, and alarms. It also automatically selects the checkbox in the **Points of Interest** column. To search for tags after configuration, see [Using Data Explorer on page 141](#).

Note: The total number of **Points Used** out of the number licensed for the project is shown in the upper-right corner of **Networks Explorer**.

- c. For all points that are to be trended, also select their checkboxes in the **Trend (his)** column.

5. Select **Update Profile & Assign**.

Note: A list of all the devices that use this profile appears in an **Assign profile** window.

6. Select the checkboxes next to the devices that you want to assign this edited profile to.
7. Select **Assign to Devices**.

Note: **Regenerating Points** will appear at the bottom and will revert to an **Assign Profile** button when the process completes. It is okay to leave the page during the process. In the network's device list, a **spinning gear icon** will appear under **Actions** until the device profile has regenerated.

Editing Device Details

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Edit Device Details

1. Go to **Networks Explorer** .
2. Select view network  from the row of the network that the device belongs to.
3. Select Edit Device  (from the row of the device that you want to edit), which makes an **Edit [Device Name] Details** window appear.
4. Edit the **Device Name**, **Model Name**, **Vendor Name**, and/or **Description**.

Note: If the device is a Modbus device, you can also set a **Read/Write Delay (ms)**.

Note: **Point Read Batch (Count)** defines how many points to read at once during a single connection to a Modbus device. The default is 4. Increasing **Point Read Batch (Count)** decreases the amount of connections made to the Modbus device, which may prevent it from locking up. (If you set **Point Read Batch (Count)** to the amount of points needing read, the KMC Commander gateway will only make one connection to the device.) However, depending on the KMC Commander gateway's connection speed, increasing **Point Read Batch (Count)** may cause it to time out.

5. Select **Save**.

Note: Later selecting Refresh Device Details  for the device could cause the changes to be overwritten.

Creating a Site Topology

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Note: In **Settings**  > **Users/Roles/Groups** > **Users**, the site topology can be used to allow users to view and control some devices and not others. (See [Adding and Configuring Users on page 18](#).)

Adding a New Node to the Site Topology

1. Go to **Networks Explorer** , then **Site Explorer**.
2. Select **Add New Node**, which opens the **Add New Node** window.
3. From the **Type** dropdown menu, choose whether the topology node is for a **Site**, **Building**, **Floor**, **Zone**, **Virtual Device**, or **Virtual Point**.

Note: For **Virtual Device** details, see [Creating a Virtual Device on page 48](#). For **Virtual Point** details, see [Creating a Virtual Point on page 48](#).

4. Enter a **Name** for the node.

Note: You can edit the node's name later by selecting it, then selecting **Edit**.

5. Select **Add**.
6. Drag and drop items to reflect the site's hierarchy.

Note: Devices can be dragged directly under a new building, floor, or zone. Zones are under floors, floors are under buildings, and buildings are under sites. A green check mark (instead of a red NO symbol) appears when dragging items onto possible locations.

Editing a Node's Properties (Area)

1. Go to **Networks Explorer** , then **Site Explorer**.

2. Select the node, then select **Edit Properties** (which appears on the node's right) to open the **Edit [Node Type] Properties** window.
3. Select the **Unit of Measure** dropdown menu, then choose **Square Feet** or **Square Meters**.
4. Enter the **Area** of the space represented by the node.
5. Select **Save**.

Creating a Virtual Device

A virtual device can contain a selection of points copied from a physical device. This is helpful if a device has many points (such as a JACE), but you want to monitor closely and/or control only a portion of them.

1. Go to **Networks Explorer** , then **Site Explorer**.
2. Select **Add New Node** to open the **Add New Node** window.
3. From the **Type** dropdown menu, select **Virtual Device**.
4. From the **Select Device** dropdown list, select the physical device that you want to copy points from for your virtual device.

Note: You can narrow the list of devices to choose from by typing in the dropdown list selector.

5. Select the checkboxes next to the points you want to copy to your virtual device.
6. Enter a **Name** for the virtual device.
7. Select **Add**.

Note: You may need to scroll down to see the **Add** button.

Creating a Virtual Point

Note: Virtual points are an advanced feature requiring knowledge of JavaScript. See [Virtual Point Program Examples on page 49](#).

1. Go to **Networks Explorer** , then **Site Explorer**.
2. Select **Add New Node** to open the **Add New Node** window.
3. From the **Type** dropdown menu, select **Virtual Device**.
4. From the **Select Device** dropdown list, select the device.

Note: You can narrow the list of devices to choose from by typing in the dropdown list selector.

5. From the **Select Point** dropdown list, select the point.

Note: You can narrow the list of points to choose from by typing in the dropdown list selector.

6. Edit the JavaScript program in the text box.

Note: For guidance, see [Virtual Point Program Examples on page 49](#).

7. Enter a **Name** for the virtual point.

8. Select **Add**.

Virtual Point Program Examples

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Virtual Points

Virtual points enable building of complex logic on top of existing points in the system without the creation of additional points or complex control code on devices. A simple JavaScript function is executed on each update of the source point(s) and can produce one or more outputs for the virtual point. Virtual points are ideal for unit conversion, computing periodic averages or sums, or for running more advanced application-specific logic.

```
function run(device, point, latest, state, emit, toolkit){  
  /*  
    device  
  */  
}
```

Term from JavaScript program	Description
function run ()	Takes in arguments (for example: point , device , etc.) and executes them each time the point is updated.
point	A JSON object that has properties, such as point.tags, which reflect Project Haystack. Examples: • point.tags.curVal (the current value) • point.tags.his (a boolean indicating whether or not the point is trended). Note: Examine the point object's available properties using Using Data Explorer on page 141.

Term from JavaScript program	Description
device	Every point is associated with a device. The device scope is a JSON object that contains relevant tag values. Note: For the data structure, please search for the device in Using Data Explorer on page 141.
latest	A JSON object with the following keys: • v: (the current value of the point, otherwise referred to as curVal) • t: (timestamp)
emit	Allows you to add to the trend value. You can pass the following: • v: (the current value of the point, otherwise referred to as curVal) • t: (timestamp)
state	An empty JSON object that can be used to save information.
toolkit	A set of JavaScript libraries, including: • Moment (a data and time utility library) • Lodash (a modern JavaScript utility library delivering modularity, performance, and extras)

Examples

Estimating Power

```
function run(device,point, latest, state, emit, toolkit){
  emit({
    t: latest.t,
    v: latest.v*115
  })
}
```

The first line contains variables coming into the function. For instance, **latest** is a variable containing the current time and value of the source point.

The second line emits variables out of the function.

latest.v is the value read from the real point.

v is the value you want the virtual point to be. This example is creating a rough estimate of power. The real point is measuring current. The virtual point will be **115** times the current reading.

The time is **t**.

The **emit** argument is a JSON object, which is a way of expressing **name:value** pairs. You can separate each pair onto its own line. Each **name:value** pair is separated by a comma.

The colon (:) is similar to an equal sign, so the name **t** is being set to **latest.t**. The value will typically be a calculation.

Binary Virtual Point to Indicate an Analog Point is Too High

```
function run(device, point, latest, state, emit, toolkit){
  emit({
    t: latest.t,
    v: latest.v > 80
  })
}
```

Continuous Sum (Sigma)

The sigma function sums all the values over time. Here we use state to persist sum and add whenever a point is updated.

```
function run(device, point, latest, state, emit, toolkit){
  // Compute the continuity of all current values (Sigma Function)
  var sigma = 0;

  if(state.sigma){
    sigma = state.sigma;
  }

  sigma+= latest.v;

  emit({
    v: sigma,
    t: toolkit.moment().valueOf()
  });
}
```

Fahrenheit to Celsius

Here is a run function that applies the Fahrenheit to Celsius formula to the latest value:

```
function run(device, point, latest, state, emit, toolkit){
// Get the latest.v point in Fahrenheit and convert to Celsius;
var c = (latest.v - 32) * (5/9);
emit({
    v: c,
    t: toolkit.moment().valueOf()
});
}
```

Celsius to Fahrenheit

Here is a run function that applies the Celsius to Fahrenheit formula to the latest value:

```
function run(device, point, latest, state, emit, toolkit){
// Get the latest point in Celsius and convert to Fahrenheit;
var f = (latest.v *(9/5)) + 32;
emit({
    v: f,
    t: toolkit.moment().valueOf()
});
}
```

Weekly Average

Here is a run function that computes the average of the values updated for a week (Sunday-Saturday):

```
function run(device,point, latest, state, emit, toolkit){
    // average
    if(state.sum == null)
        state.sum = 0;
    if(state.num == null)
        state.num = 0;
    if(state.t == null)
        state.t = toolkit.moment(new Date()).startOf('week');
    state.num++;
    state.sum += latest.v;
    // only emit once we've passed the end of a day
    if(toolkit.moment(latest.t).startOf('week')!=toolkit.moment
(state.t).startOf('week')){
        emit({t: toolkit.moment(state.t).endOf('day'), v: state.sum/state.num});
        state.t = null;
        state.num = null;
        state.sum = null;
    }
}
```

Finding and Deleting Orphan Nodes

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Find and Delete Orphan Nodes

Sometimes in the processes of adding or removing devices or points and creating cards, you end up with:

- devices you are no longer using that have lost a network reference
- points you are no longer using that have lost a device reference

Collectively these devices and points are called *orphan nodes*.

To find and delete orphan nodes:

1. Go to **Networks** , then **Orphan Nodes**.
2. From the option buttons, choose either **Devices** or **Points**.
3. Select all orphaned nodes using the select all checkbox, or select specific points that you want to delete.
4. Select **Delete Nodes**.

Note: The nodes will delete immediately. No confirmation is required.

Dashboards and Their Elements

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Dashboards can hold cards, decks, canvases, and report modules. The initial home screen will be blank before adding a dashboard. Once you add a dashboard, you can add instances of cards, decks, and canvases.

Cards are the primary means to visualize network data and control equipment from a web browser. Cards allow users to change setpoints and view equipment point values. To be able to command a point from a card, the point must be made commandable (under the Type column) in the device profile (for example, Analog > Command). You do not have to configure points you don't want to use.

Decks are an optional method of organizing the cards (such as the most critical cards or all the cards related to a particular floor). Decks can show a carousel of the included cards.

Canvases are creative spaces to arrange points and/or zone shapes (both with customizable colors and opacity) on a background image uploaded from your computer. Displaying live point values on equipment graphics and floor plans are typical uses.

After configuring report settings in **Reports** , you can add an instance of a *report module* or a *report card* to a (non-global) dashboard to display the report.

Dashboards and their elements are specific to user logins. Decks added by a system administrator or technician for a site will be available to add to the dashboard of that customer. This is a convenient way for a customer to create their own dashboard without needing to create each card from scratch.

In the KMC license server, KMC can also add a customer image URL to the license. The logo or other image will then display to the left of the project name on the dashboard. (To use this feature, supply KMC Controls with the image's URL address.)

Adding and Configuring Dashboards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Adding a New Dashboard

1. Select **Dashboards** , which opens the dashboard selector sidebar.
2. Select one of the options (at the bottom of the dashboard selector):
 - **Add Dashboard** —creates a standard dashboard, on which you can display information only from the project that the dashboard belongs to.
 - **Add Global Dashboard** —creates a global dashboard, on which you can display information from any project that you have access to, not just from the project to which the global dashboard belongs. The dashboard will have globe icon  to indicate that it is a global dashboard.



Caution: Currently, point override display and default write values will use the current project's settings rather than the individual projects' settings. (See [Display Point Override on page 10](#), [Default Manual Write Priority on page 15](#), and [Manual Write Timeout on page 15](#).) If individual projects' settings differ, take care when making point override changes or interpreting an override warning on a global dashboard.

Note: A dashboard preview named "New Dashboard" appears in the dashboard selector and the new, empty dashboard displays in the viewing window. See [Renaming a Dashboard on page 58](#) for how to change the name.

Setting a Dashboard Preview Image

1. Go to the dashboard that you want to set the preview image for.
2. Select the gear icon  (next to the dashboard's name), which makes the dashboard settings menu appear.
3. Select **Set Preview Image**.

Note: An **Upload For [dashboard name]** window appears.

4. Select **Choose file**.
5. Find and open the image file from your computer that you want to be the preview image.

Note: The recommended image dimensions are 550px by 300px. It must be less than 5 MB. An image optimized to the smallest file size possible (without losing the quality needed) is recommended. Accepted file types are .png, .jpeg, and .gif.

6. Select **Upload**.

Setting a Dashboard's Width

When a dashboard is added, its width is the [Fixed Dashboard Width on page 10](#) set in **Settings**  > **Project Settings**.

Note: Hover over the columns  icon to find out the number of columns **Fixed Dashboard Width** is set to. If no columns  icon, **Fixed Dashboard Width** is set to **Auto** (i.e. a responsive layout).

You can also individually set a dashboard's width. For that dashboard the individual setting will override the project-wide setting. To set a **Dashboard Width**:

1. On the dashboard that you want to set the width for, select **Configure Dashboard** .
2. Select **Dashboard Width**, which opens a **Set Dashboard Width** window.
3. From the dropdown menu, select the desired number of columns, or enter the number.

Note: A column is the width of one medium-sized card (for example, one Weather card).

4. Select **Save**.

Note: Hovering over the columns  icon will show the number of columns set.

Note: A left-right scroll bar will appear on narrower screens and browser windows.

Changing the Dashboard Refresh Interval

To change the **Refresh Interval** at which elements on *all* dashboards are updated with Cloud data:

1. With a dashboard displayed, select **Configure Dashboard** .
2. Select **Refresh Interval**, which makes the **Set Refresh Time** window appear.
3. Select the desired interval from the drop-down menu.

Note: **Refresh Interval** is the interval at which dashboards fetch data from the Cloud. It does not change the interval at which devices are polled for data, which is set in **Settings**  > **Protocols** > [Point Update Wait Interval \(Minutes\)](#) on page 15.

4. Select **Save**.

Setting a Dashboard as the Homepage

When a dashboard is set as the homepage, it is the first dashboard that appears after logging in.

1. Go to the dashboard that you want to make the homepage.
2. Select the gear icon .
3. Select **Set as homepage**.

Selecting a Dashboard to View

1. Select **Dashboards** , which makes the dashboard selector sidebar appear.

Note: For users with **Admin** permissions (see [Configuring Roles on page 23](#)), there is a switch at the top of the selector. Toggle the switch to either **Showing only your dashboards** or **Showing all dashboards** (for the project).

2. Find and select the name or preview of the dashboard that you want to view.

Note: You can narrow the list by typing in **Search dashboards**.

Note: The dashboard appears in the viewing area to the right.

Making a Copy of a Dashboard

1. Go to the dashboard that you want to make a copy of.

2. Select the gear icon .
3. Select **Make a copy**.

Note: The copy is made and is displayed in the viewing area. The copy has the same name as the original plus a number in parenthesis at its end. See [Renaming a Dashboard on page 58](#) for how to change the name.

Sharing Dashboards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

1. With the dashboard that you want to share displayed in the viewing window, hover over the dashboard's name.
2. Select the gear icon  that appears.
3. Select **Share**, which opens the **Share dashboard** window.

Note: You can select other dashboards to share besides the one currently displayed by choosing them from the **Select dashboard** dropdown list.

4. Select the checkboxes of the **Users** to whom you want to give **Read-only** access, **Write Access**, or **Share a Copy** of the dashboard.

Note: See [Types of Sharing on page 57](#) for details of each option.

5. Select **Submit**.

Types of Sharing

Read-only

Read-only access allows the other users to see the dashboard, but not modify cards or decks. Any changes made to the dashboard from your account can be seen automatically by the other users from their accounts. From your account, a group icon will show next to the name of the dashboard. Hovering a cursor over the icon displays a message stating the number of users with whom the dashboard is shared. From the other users' accounts, an eye icon will show next to the name of the dashboard, indicating it is read-only.

Note: While other users will not be able to modify the dashboard's cards, *setpoints* on those cards may still be editable depending on a user's role.

Write Access

Write Access allows the other users to both see and edit the dashboard. Any changes made to the dashboard from your account can be seen by the other users from their accounts. Likewise, any changes made to the dashboard from the other user's accounts can be seen from your account. A group icon will show next to the name of the dashboard when viewed from all users' accounts. Hovering a cursor over the icon displays a message stating the number of users with whom the dashboard is shared.

Note: It is advised that not more than one user customize a card at the same time. If multiple users are in a card's customize mode at once, the user who exits customize mode last (by clicking the pencil icon) will overwrite the other user(s) changes.

Share Copy

Share Copy makes “snapshot” copies of the dashboard as it is currently set up and shares those copies with the other users, which they can then customize as needed. The original dashboard and its copies are not connected in any way. Any subsequent changes you make to the original dashboard will not be reflected in the copies shared with the other users. Likewise, any subsequent changes that the other users make to their copies will not be reflected elsewhere.

Modifying (and Deleting) Dashboards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Renaming a Dashboard

A dashboard can be renamed from either the dashboard selector or when it is displayed in the viewing window.

From the Dashboard Selector

1. If the dashboard selector is not already open, select **Dashboards**  to open it.
2. Select the **gear icon** in the dashboard preview of the dashboard you want to rename.
3. Select **Rename**.

From the Viewing Window

1. Go to the dashboard that you want to rename.
2. Select the gear icon .
3. Select **Rename** from the menu that appears.
4. Enter a new **Dashboard name**.
5. Select **Submit**.

Rearranging Cards and Decks on a Dashboard

1. In **Dashboards** , select **Edit Layout** (in the upper-right of corner of the dashboard).

Note: This causes the grip icon  to appear in the upper-right corner of cards and decks.

2. Grab (select and hold) a card or deck that you want to move by its grip .
3. Drag the card or deck over to where you would like it to be.

Note: The other cards automatically rearrange to make room for the card.

4. Drop the card or deck in its new location.
5. Keep rearranging cards and decks until the layout is the way you would like it.
6. Select **Save Layout**.

Deleting a Dashboard

1. Go to the dashboard that you want to delete.
2. Select the gear icon .
3. Select **Delete**.
4. Select  (Confirm Delete).

Creating and Adding Cards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Creating and Adding Cards

For maximum performance, if the desired number of cards (depending on complexity) exceeds 12, make multiple dashboards with fewer cards on each dashboard. For example, make several dashboards for system-level views and other dashboards for equipment-level details.

Creating a Custom Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Custom Cards

If one of the standard card types does not meet the application need, you can create a simple **Custom card**, which shows values in up to 10 slots.

Creating the Custom Card

Access the Custom Card Staging Area

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Custom card** (if not already selected) from the card type options at the left.

Select Points

For each slot that you want to fill with a point:

1. Select **Select Point**, which makes the **Device** list and **Point Selector** appear.

Note: The **Point Slot** tab is selected by default.

2. Locate and choose the point.

Note: If creating on a global dashboard  , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

Add Text Slots (Optional)

1. Select **Select Point**.

Note: The **Device** and **Point Selector** appear, because the **Point Slot** tab is selected by default.

2. Select **Text Slot**, which switches to a text editor tab.
3. Type and format text and/or hyper-linked text, as you would in a simple word processor.
4. Select **Save**.

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Creating a KPI Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About KPI Cards

KPI (Key Performance Indicator) cards are smaller than other cards and can track a **point** in a particular device or track a **metric**. Metrics are, for example, the BTU rate or electric power for an entire floor, zone, building, or site, based on the topology set up in **Network Explorer**  > **Site Explorer**. KPI metrics are based on area. **Edit Properties** in **Site Explorer** provides fields to enter area values and units (see [Editing a Node's Properties \(Area\)](#) on page 47).

Creating the KPI Card

Access the KPI Card Staging Area

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **KPI card** from the card type options at the left.

Select a Point

1. Select **+**, which makes the **Device** list and **Point Selector** appear.
2. Locate and choose the point.

Note: If creating on a global dashboard , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile](#) on page 45). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

Add Status Colors

See [Adding Status Colors on page 62](#) for details.

Add Text Slots (Optional)

1. Select **Select Point**.

Note: The **Device** and **Point Selector** appear, because the **Point Slot** tab is selected by default.

2. Select **Text Slot**, which switches to a text editor tab.
3. Type and format text and/or hyper-linked text, as you would in a simple word processor.
4. Select **Save**.

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Adding Status Colors

When status colors are configured, a color-coded status bar displays on the left edge of the card's point slot. You can configure the status color to change depending on point's current value.

Using Premade Color Sets

1. Select Add colors  (on the point slot's left), which makes a window appear.
2. Choose a **Color Set** from the dropdown menu.
3. Enter the **Min value** and **Max value**.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

4. If you want this color configuration to apply to the text as well, select the **Apply color to text** checkbox.
5. Select **Save** to apply the status color configuration to the point.

Using a Custom Color Set

1. Select Add colors  (on the point slot's left), which makes a window appear.
2. From the **Color Set** dropdown menu, select **Custom**.
3. Enter the **Min value** and **Max value**.

Note: To add intermediate values, select the **+** (Add intermediate value). Then enter the new **Intermediate value**.

4. Select the thumbnails below the color spectrum, which opens a color palette.
5. Do one of the following to choose a color:
 - Use the color slider and move the selection circle.
 - Enter the **HEX** color code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
6. Do one of the following to change the opacity:
 - Use the opacity slider.
 - Change the seventh and eighth digits of the **HEX** code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
7. If you want this color configuration to apply to the text as well, select the **Apply color to text** checkbox.
8. Select **Close**.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

9. Select **Save** to apply the status color configuration to the point.

Creating a KPI Gauge Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About KPI Gauge Cards

KPI (Key Performance Indicator) gauge cards are smaller than other cards and track a **point** in a particular device or track a **metric**. KPI *gauge* cards display a number (like KPI cards), plus an animated gauge graphic. Metrics are, for example, the BTU rate or electric power for an entire floor, zone, building, or site, based on the topology set up in Network Explorer's Site Explorer. KPI metrics are based on area. Fields to enter area values and units are found within **Networks Explorer**  > **Site Explorer**. See [Editing a Node's Properties \(Area\) on page 47](#) for details.

Creating the KPI Gauge Card

Access the KPI Gauge Card Staging Area

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.

3. Select **KPI gauge** from the card type options at the left.

Select a Point

1. Select **Select Point**, which makes the **Device** list and **Point Selector** appear.
2. Locate and choose the point.

Note: If creating on a global dashboard , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

Configure the Gauge

1. Choose a **Color Range** for the gauge.

Note: Default is a white to orange gradient.

2. Choose the **Gauge Type**: **Gauge** or **Gauge with Needle**.
3. Enter the gauge's:
 - **Min** (minimum) value.
 - **Lower Middle** value (only for a gauge with a needle).
 - **Upper Middle** value (only for a gauge with a needle).
 - **Max** (maximum) value.

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Configuring the Area

Fields to enter area values and units are found within **Networks Explorer**  > **Site Explorer**. See [Editing a Node's Properties \(Area\) on page 47](#) for details.

Creating a Trend Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Trend Cards

Trend cards display point values over time on a graph. Graph information can be displayed by **Day**, **Week**, or **Month**. Slider bars below the graph allow zooming in on particular sections. Placing the cursor on the line shows information about that point at that time. The present values of points are shown in slots below the graph. Any commandable points (for example, a setpoint) can be written to using the card.

When a trend card is sized to **Wide**, **Large**, or **Extra Large**, the data can be viewed in **Realtime**, or by **Daily (Avg)**, **Weekly (Avg)**, or **Monthly (Avg)**.

Creating the Trend Card

Access the Trend Card Staging Area

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Trend** from the card type options at the left.

Select Points

For each slot that you want to fill with a point:

1. Select **Select Point**, which makes the **Device** list and **Point Selector** appear.

Note: The **Point Slot** tab is selected by default.

2. Locate and choose the point.

Note: If creating on a global dashboard  , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

Add Text Slots (Optional)

1. Select **Select Point**.

Note: The **Device** and **Point Selector** appear, because the **Point Slot** tab is selected by default.

2. Select **Text Slot**, which switches to a text editor tab.
3. Type and format text and/or hyper-linked text, as you would in a simple word processor.
4. Select **Save**.

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Creating a Thermostat Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Thermostat Cards

Thermostat cards display values, such as temperature, humidity, and CO₂, as well provide control of setpoints and other commandable (writable) points. Selecting the heating setpoint, cooling setpoint, or a writable slot on the card allows changing of the value, with a specific write priority and timeout.

Creating the Thermostat Card

Access the Thermostat Card Staging Area

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Thermostat** from the card type options at the left.

Select Points

For each slot that you need configured:

Note: In most cases, the central slot, heating slot, and cooling slot should be configured.

1. Select the slot on the card preview (such as **Select Point**), which makes the **Device** list and **Point Selector** appear.
2. Locate and choose the point that corresponds to the type of slot selected.

Note: If creating on a global dashboard , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

Add Text Slots (Optional)

1. Select **Select Point**.

Note: The **Device** and **Point Selector** appear, because the **Point Slot** tab is selected by default.

2. Select **Text Slot**, which switches to a text editor tab.
3. Type and format text and/or hyper-linked text, as you would in a simple word processor.
4. Select **Save**.

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Creating a Weather Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Weather Cards

Weather cards show the current outside air temperature, relative humidity, and weather conditions on their top half, and a four-day forecast on the bottom.

Before Beginning

In **Settings**  > **Weather**:

- Add weather stations.
- Select the default units (Fahrenheit or Celsius) to display on weather cards.

Note: See [Configuring Weather Settings on page 26](#) for details.

Creating the Card

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Weather** from the card type options at the left.
4. Choose a **Weather Station** from the dropdown list.

Note: Initially, **Card title** is the same as **Weather Station** (the city's name). However, you can change the card's name directly from the dashboard later.

5. Select **Add**.
6. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Note: There is only one **Size Type (Medium)** for weather cards.

Creating a Web Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Web Cards

Web cards can display webpages. The webpage must be HTTPS with a public URL (no on-premise IPs), and the site must allow HTML Inline Frame (iframe) elements.

Applications include:

- Documents
- Live, cloud-based camera feeds

Note: This does not include local CCTV camera feeds.

- Node-RED dashboards
- Videos

Note: For a video on YouTube, use the address within the iframe tag found within Share > Embed below the video (for example, https://www.youtube.com/embed/_f3ijEWDv8k). A URL taken directly from the YouTube browser window will not work.

- Weather radar
- Webpages with forms for submission

Creating the Card

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Web** from the card type options at the left.
4. Enter a **Card title**.
5. Choose a default **Size Type** from the dropdown menu.
6. Enter a valid **Web URL**.

Note: See [About Web Cards on page 69](#) for guidance about valid URLs.

7. Select **Validate URL**.

Note: If the URL is valid, a notification that reads "[URL] can be embedded" will appear briefly. If it is invalid, the message will read, "Please make sure this is an https URL with a valid source, and the X-Frame-Options header is set to allow".

8. Select **Add**.
9. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Creating a Text Editor Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Text Editor Cards

Text Editor cards allow you to compose and display text such as you would in a simple note app.

Examples of applications include displaying:

- Links to PDF files.
- Links to saved report settings (see [Linking to a Report on page 135](#)).
- Equipment instructions.
- Cautionary warnings.
- User manuals.
- Contact info.

Creating the Card

1. With the dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Text Editor** from the card type options at the left.
4. Enter a **Card title**.
5. Choose a default **Size Type** from the dropdown menu.
6. Compose text on the card.

Note: You can compose text on the card now, or directly from the dashboard later.

Note: See [Composing Text on page 71](#) for details.

7. Select **Add**.
8. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Composing Text

Accessing the Card's Edit Mode

1. Move over the space to the right of the card's title.
2. Select the gear icon  , which enables the card's Edit Mode.

Typing, Formatting, and Saving Text

1. Type and format the text as you would in a simple word processor.
2. **Close Edit Mode**, which saves your changes.



Caution: Close Edit Mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Creating Links to Web URLs

1. Highlight the text that you want to make into a hyperlink.
2. Select the link icon  .
3. Copy and paste into **Enter link** the web URL that you want to link to.
4. Select **Save**.
5. **Close Edit Mode**, which saves your changes.



Caution: Close edit mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Creating a Report Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Report Cards

After configuring a report setting in **Reports**  , you can display the report on a (non-global) dashboard using a **Report Card**.

Alternatively, you can add a **Report** module. (See [Adding a Report Module on page 92.](#)) **Report** modules can easily switch between report settings. However, unlike a **Report Card**, a **Report** module always spans the entire width of a dashboard.

Creating the Report Card

Access the Report Card Staging Area

1. With the (non-global) dashboard that you want to add the card to displayed, select **Add Instance**.
2. Select **Card**, which opens the card staging area.
3. Select **Report Card** from the card type options at the left.

Select a Report Setting

From the **Select Report** dropdown list, select the setting of the report that you want to display.

Note: The listed report settings are configured in **Reports** . (See [Managing Reports on page 124.](#))

Title and Size

1. Enter a **Card title**.
2. Choose a default **Size Type** from the dropdown menu.

Add to the Dashboard

1. Select **Add**.
2. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Duplicating a Card Across Devices

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Duplicate a Card

If several devices use the same profile, you can create a card for one of the devices, then automatically duplicate that card for the other devices.

1. Hover at the top edge of the device's card that you want to duplicate for other devices.
2. Select the More icon  on the toolbar that appears.
3. Select **Duplicate Card**.

Note: A list of all the other devices that share the same profile appears to the right.

Note: If no other devices also have this profile, a message will appear to the right. Assign this device's profile to other devices. (See [Assigning Device Profiles on page 43.](#))

Note: If this card contains more than one device's points, it can not be automatically duplicated. Create each card manually. (See [Creating and Adding Cards on page 59.](#))

4. Check the boxes next to the devices that you want to duplicate this card for.
5. Leave the **Naming Convention** as is, or modify it.

Note: <Device> will automatically insert each device's name into its card title.

6. Select **Duplicate**.

Note: The cards are automatically created and added to the bottom of the dashboard.

Modifying Cards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Editing a Card's Title

1. Move over the space to the right of the card's title.
2. Select the More icon  on the toolbar that appears.
3. Select **Rename Card**.
4. Edit the **Card Title** as needed.
5. Select **Submit**.

Changing or Adding Points on a Card

1. On a card with configurable device points, hover near the upper-right corner, which causes a toolbar to appear.
2. Select the gear icon , which opens the card's **Edit Mode**.
3. Select the point slot that you want to change, which makes a **Device** list and **Point Selector** appear.
4. Locate and choose the needed point.

Note: If creating on a global dashboard , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

5. **Close Edit Mode.**

Reconfiguring the Area, Range, and Color of a KPI Gauge Card

1. Move over the space to the right of the KPI gauge card's title.
2. Select the More icon  on the toolbar that appears.
3. Select **Configure**.
4. Modify the **Area**, **Min**, **Max**, and **Color Range** as needed.
5. Select **Submit**.

Changing the Weather Station Displayed by a Weather Card

1. Move over the space to the right of the Weather card's title.
2. Select the More icon  on the toolbar that appears.
3. Select **Edit Weather Station**, which causes a list to appear to the right.
4. Choose the weather station that you want the card to display.

Changing the Webpage Displayed by a Web Card

1. Move over the space to the right of the web card's title.
2. Select the More icon  on the toolbar that appears.
3. Select **Set Web URL**, which opens the **Edit Web URL** window.
4. Enter the **Web URL** that you want the card to display.
5. Select **Validate**.

Note: If the URL is valid, **Validate** will change to **Save**. If the URL is invalid, a message will appear briefly that reads, "This website is blocking Commander. Please make sure this is an https URL with a valid source, and the X-Frame-Options header is set to allow." The website may be blocking Commander or the text entered for **Web URL** may simply have a typographical error.

6. Select **Save**.

Hiding and Showing Trend Lines

On a Trend card, hide/ show a trend line by toggling on/ off the dot matching the color of the trend line that you want to hide/ show.

Note: The colored dots are in front of the point names (in the point slots) corresponding to the trend lines. If the point slots are not visible, hover over the area next to the card name and select the resize arrows  that appear.

Composing Text on a Text Editor Card

Accessing the Card's Edit Mode

1. Move over the space to the right of the card's title.
2. Select the gear icon , which enables the card's Edit Mode.

Typing, Formatting, and Saving Text

1. Type and format the text as you would in a simple word processor.
2. **Close Edit Mode**, which saves your changes.



Caution: Close Edit Mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Creating Links to Web URLs

1. Highlight the text that you want to make into a hyperlink.
2. Select the link icon .
3. Copy and paste into **Enter link** the web URL that you want to link to.
4. Select **Save**.
5. **Close Edit Mode**, which saves your changes.



Caution: Close edit mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Using Cards

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Writing to a Point

Using the simplified method

1. Select the setpoint slot on the card, which opens a window titled with the setpoint's name.
2. Enter the new value for the setpoint.
3. Select **Write Priority [Default]**.

Note: The priority given here is the [Default Manual Write Priority on page 15](#), configured in **Settings**  **> Protocols**.

Note: The value will be written for the duration of the [Manual Write Timeout on page 15](#) (default **None**), configured in **Settings**  **> Protocols**.

Using Advanced Settings

1. Select the setpoint slot on the card, which opens a window titled with the setpoint's name.
2. Enter the new value for the setpoint.
3. Select **Show Advanced Settings**, which expands to allow you to:
 - Select a **Write Priority** from the dropdown menu.
 - Select a **Write Timeout** from the dropdown menu.

Note: **Write** should be selected (by default) for **Write Value or Clear Slot**.

Note: A history of the current and previous 10 reads of the priority array displays below. Scroll to the right to view all 10. The interval of the time stamps is partly determined by the [Read Priority Array Wait Interval \(Minutes\) on page 15](#).

4. Select **Write Priority _**.

Note: It may take a minute for the point on the *device* to change to the new value so that the card shows the change. See also the [Read Time After Point Writes \(Seconds\) on page 10](#), configured in **Settings**  **> Protocols**.

Clearing a Priority

1. Select the setpoint slot on the card, which opens a window titled with the setpoint's name.
2. Select **Show Advanced Settings**.
3. For **Write Value or Clear Slot**, select **Clear**.
4. From the **Clear Priority** dropdown menu, choose the priority that you want to clear.

Note: A history of the current and previous 10 reads of the priority array displays below. Scroll to the right to view all 10. The interval of the time stamps is partly determined by the [Read Priority Array Wait Interval \(Minutes\)](#) on page 15.

5. Select **Clear Priority** .

Note: It may take a minute for the point on the *device* to clear the value so that the card shows the change. See also the [Read Time After Point Writes \(Seconds\)](#) on page 10, configured in **Settings**  **> Protocols**.

Flipping to the Back of a Card

Note: You can flip Custom cards, KPI Gauge cards, and Thermostat cards to show more information from a device and command additional points.

1. Move over the bottom edge of the card.
2. Select the **Flip to back** that appears.

Note: The rows show the present values of all points of interest on that device. Any row that is shaded is a selectable and commandable point. When done, select **Flip to front**.

Rearranging Cards and Decks on a Dashboard

1. In **Dashboards** , select **Edit Layout** (in the upper-right of corner of the dashboard).

Note: This causes the grip icon  to appear in the upper-right corner of cards and decks.

2. Grab (select and hold) a card or deck that you want to move by its grip .
3. Drag the card or deck over to where you would like it to be.

Note: The other cards automatically rearrange to make room for the card.

4. Drop the card or deck in its new location.
5. Keep rearranging cards and decks until the layout is the way you would like it.
6. Select **Save Layout**.

Favoriting a Card

Prerequisites

If you favorite a card, it is added to a **Favorites** deck. Therefore, you must first have a deck titled "Favorites" for  (**Favorite Card**) to work. (See [Finding a Deck in the Deck Library](#) and [Using the deck creation area on page 80.](#))

Adding a Card to the Favorites Deck

1. Hover over the upper-right corner of the card.
2. Select the circle  that appears, which selects the card.
3. Select  (**Favorite Card**).

Note: If a deck titled "Favorites" exists (see [Finding a Deck in the Deck Library](#)), it is added there automatically. If it doesn't exist, an error message briefly appears. Though the message says "Please create a dashboard titled 'Favorites'", you must create a *deck* titled "Favorites" (see the [Prerequisites on page 77](#)).

Hiding and Showing Trend Lines

On a Trend card, hide/ show a trend line by toggling on/ off the dot matching the color of the trend line that you want to hide/ show.

Note: The colored dots are in front of the point names (in the point slots) corresponding to the trend lines. If the point slots are not visible, hover over the area next to the card name and select the resize arrows  that appear.

Composing Text on a Text Editor Card

Accessing the Card's Edit Mode

1. Move over the space to the right of the card's title.
2. Select the gear icon , which enables the card's Edit Mode.

Typing, Formatting, and Saving Text

1. Type and format the text as you would in a simple word processor.
2. **Close Edit Mode**, which saves your changes.



Caution: Close Edit Mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Creating Links to Web URLs

1. Highlight the text that you want to make into a hyperlink.
2. Select the link icon .
3. Copy and paste into **Enter link** the web URL that you want to link to.
4. Select **Save**.
5. **Close Edit Mode**, which saves your changes.



Caution: Close edit mode before navigating away from the dashboard. Navigating away before closing Edit Mode discards any changes.

Taking Actions From a Report Card

See [Using a Report on page 136](#).

Deleting a Card

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Directly from the Dashboard

You can delete a single card or multiple cards at once using the direct method.

1. Hover over the upper-right corner of the card.
2. Select the circle  that appears, which selects the card.
3. Repeat for any other cards that you want to delete.
4. Select delete  on the toolbar that appears at the bottom of the application window.
5. Select **Confirm**.

Using a Card's Menu

You can delete one card at a time using this method.

1. Hover over the upper-right corner of the card.
2. Select the More icon  that appears.
3. Select **Delete**.
4. Select Confirm Delete  .

Creating and Adding Decks

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Adding Cards to a New Deck

After [Creating and Adding Cards on page 59](#) to a dashboard, you can add instances of those cards to a deck.

Note: See also [Adding a Card to an Existing Deck on page 81](#).

From a dashboard directly

1. Hover over the upper-right corner of a card that you want to add to a new deck.

2. Select the circle  that appears, which selects the card.
3. Repeat step 2 for any other cards that you want to add to the same deck.
4. Select  (**Add Cards to Deck**), which opens the **Add card(s) to decks** window.
5. Select **+ New Deck** (at the bottom of the list, which makes the text editable).
6. Replace the text with a name for the new deck.
7. Press **enter**, or select an area outside of the text box.

Note: The checkbox for the new deck is automatically selected for you.

8. Select **Add**.

Note: The new deck appears at the bottom of the dashboard. It is also added automatically to the deck library.

Note: You can set the default deck view mode in **Settings**  > **Project** > **Dashboard**. See [Dashboard Deck Mode on page 9](#) for details.

Using the deck creation area

1. With the dashboard that you want to add the deck to displayed, select **Add Instance**.
2. Select **Deck**.
3. Switch the toggle at the upper-left to **Create new deck**.
4. Choose the cards that you want to add to the new deck by hovering over a card's upper-right corner, then selecting the circle  for it.
5. Select **Continue**.
6. Enter a **Deck name**.
7. Select **Submit**.

Note: The new deck appears at the bottom of the dashboard. It is also added automatically to the deck library.

Note: You can set the default deck view mode in **Settings**  > **Project** > **Dashboard**. See [Dashboard Deck Mode on page 9](#) for details.

Adding a Deck from the Deck Library to a Dashboard

Once a deck is created, it is added automatically to that dashboard and the deck library. Even if the deck is later deleted from the dashboard, it still exists in the deck library so that you can later add it to the same or other dashboards.

1. With the dashboard that you want to add the deck to displayed, select **Add Instance**.

2. Select **Deck**, which opens the deck selection area in the **Select existing decks** view.
3. Choose the deck that you want to add by selecting the circle  for it.

Note: You can add more than one deck at a time by choosing multiple decks.

4. Select **Add**.
5. Choose to either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Note: You can set the default deck view mode in **Settings**  > **Project** > **Dashboard**. See [Dashboard Deck Mode on page 9](#) for details.

Modifying Decks

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Rearranging Cards in a Deck

1. Go to the deck on a dashboard, or in the deck library.

Note: See [Finding a Deck in the Deck Library](#).

2. Select **Rearrange Cards** , which makes a **Rearrange cards** window appear.
3. Drag the card titles and drop them higher or lower in the list to rearrange the left-to-right order of the cards in the deck.

Note: The cards are listed top to bottom in the order that they appear left to right when the deck is in the **Expand Down** view mode. (See [Switching Between Deck View Modes on page 83](#).)

4. Select **Submit**.

Adding a Card to an Existing Deck

Note: See also [Adding Cards to a New Deck on page 79](#).

1. In **Dashboards** , hover near the upper-right corner of the card that you wish to add.
2. Select the More icon  on the toolbar that appears.
3. Select **Add to Decks**, which makes a list appear of all existing decks in the deck library.
4. Check the box next to the deck that you want to add the card to.

Note: A confirmation message appears briefly in the upper-right corner of the dashboard.

Note: You can add the card to more than one deck at once (and also remove it).

Removing a Card from a Deck

Using the direct method

1. Go to the deck on a dashboard, or in the deck library.

Note: See [Finding a Deck in the Deck Library](#).

2. Hover near the upper-right corner of the card that you want to remove.
3. Select remove/delete .

Using the card's menu

If an instance of a card is placed individually on a dashboard as well as in a deck, you can remove the deck instance using the individual instance's card menu.

1. Go to the individual instance of the card on the dashboard.
2. Hover near the upper-right corner of the card.
3. Select the More icon  on the toolbar that appears.
4. Select **Add to Decks**, which makes a list appear of all existing decks in the deck library.
5. Clear the checkbox next to the deck that you want to remove the card from.

Note: A confirmation message appears briefly in the upper-right corner of the dashboard.

Note: You can remove the card from more than one deck at once (and also add it).

Editing a Deck's Title

1. Go to the deck on a dashboard, or in the deck library.

Note: See [Finding a Deck in the Deck Library](#).

2. Select the deck's title, which makes an **Edit Deck Title** window appear.
3. Edit **Deck Title**.
4. Select **Submit**.

Using Decks

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

This section explains how to use features unique to decks. For guidance on using a deck's cards, see [Using Cards on page 75](#).

Switching Between Deck View Modes

Decks have the following view modes:

- **Perspective** (default) displays the cards in a rotatable carousel, with the central card foregrounded and the surrounding cards smaller in a shadowed background.
- **Flat** displays the cards in full size in a rotatable carousel, with the central card in full color and the surrounding cards in shadow.
- **Expand Down** displays the cards similarly to how they look when individually placed on a dashboard (all same size in full color), but grouped together into a single unit.

Note: The deck may expand down to another row, depending on the number of cards in the deck and the width of the browser window.

To switch between a deck's view modes, toggle the button in its upper-right corner (**Switch to Flat / Expand Down / Switch to Perspective**).

Note: You can set the default deck view mode in **Settings**  > **Project** > **Dashboard**. See [Dashboard Deck Mode on page 9](#) for details.

Centering a Card in a Deck

When a deck is in **Perspective** or **Flat** view mode (see [Switching Between Deck View Modes on page 83](#)), to change which card is in the center:

- Use the rotate left and right buttons  in the deck's upper-left corner.
- Click or tap the card that you want to be center, which will rotate the deck and center that card automatically.

Rearranging Cards and Decks on a Dashboard

1. In **Dashboards** , select **Edit Layout** (in the upper-right of corner of the dashboard).

Note: This causes the grip icon  to appear in the upper-right corner of cards and decks.

2. Grab (select and hold) a card or deck that you want to move by its grip .
3. Drag the card or deck over to where you would like it to be.

Note: The other cards automatically rearrange to make room for the card.

4. Drop the card or deck in its new location.
5. Keep rearranging cards and decks until the layout is the way you would like it.
6. Select **Save Layout**.

Deleting Decks

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Deleting a Deck from a Dashboard

1. With the dashboard that you want to delete the deck from displayed, select the circle  for that deck.

Note: An orange border indicates that the deck is selected and a white toolbar appears at the bottom of the browser window.

2. Select delete .

Note: After deleting a deck from a dashboard, the deck still exists in the deck library found at **Add Instance > Deck > Select existing decks**.

Deleting a Deck from the Deck Library

1. Go to the deck library by selecting **Add Instance** (in **Dashboards** ), then **Deck**.

Note: The deck selection area opens with the **Select existing decks** view (which contains the deck library) displayed.

2. Select the circle  on the deck(s) that you want to delete permanently.

Note: To avoid selecting decks for deletion that are being used on a dashboard, switch the view to **Hide Used Decks**  (if not already active). Conversely, if you wish to delete a deck that is currently in use, switch the view to **Show Used Decks** .

3. Select **Delete**.



Caution: If the deck is on other users' dashboards, it will also be deleted from there.

4. Select **Confirm**.

Adding a Canvas

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Creating a canvas involves [Accessing the Staging Area on page 84](#), [Uploading a Background Image on page 85](#), [Adjusting the Margins on page 85](#), [Adding and Styling Canvas Points on page 85](#) (separate topic), [Drawing and Configuring Canvas Zones on page 88](#) (separate topic), [Naming the Canvas on page 85](#), and [Saving the Canvas on page 85](#) to the dashboard.

Accessing the Staging Area

1. With the dashboard that you want to add the canvas to displayed, select **Add Instance**.
2. Select **Canvas**, which opens the staging area.

Uploading a Background Image

Before uploading to the canvas, size the image to the width and height that you want it to have when displayed on the dashboard. The image must also be less than 5 MB. An image optimized to the smallest file size possible (without losing the quality needed) is recommended. Accepted file types are .png, .jpeg, and .gif.

1. Select **Choose file**.
2. Find and open the image file from your computer that you want to be the background.

Adjusting the Margins

1. Select **Change card margins** .
2. Adjust the number of pixels for the **Top Margin**, **Right Margin**, **Bottom Margin**, and/or **Left Margin**.
3. Select an area outside of the **Edit Margin** window to close it.

Naming the Canvas

Now or after [Adding and Styling Canvas Points on page 85](#) and/or [Drawing and Configuring Canvas Zones on page 88](#):

1. Select **New Canvas** (the default name), which opens an edit window.
2. Enter a name for the canvas.
3. Select **Submit**.

Saving the Canvas

Select **Submit** (now or after [Adding and Styling Canvas Points on page 85](#) and/or [Drawing and Configuring Canvas Zones on page 88](#).)

Note: The canvas is added to the bottom of the dashboard.

Adding and Styling Canvas Points

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

The procedures in this topic come *after* [Adding a Canvas on page 84](#) ([Accessing the Staging Area on page 84](#), [Uploading a Background Image on page 85](#), and [Adjusting the Margins on page 85](#)), but *before* [Naming the Canvas on page 85](#) and [Saving the Canvas on page 85](#).

Adding and Arranging Points

1. Select **Add point**, which makes a **Device list** and **Point Selector** appear.
2. Locate and choose the point.

Note: If creating on a global dashboard  , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

3. Drag the point slot to where you want it to be on (or around) the background.



Caution: If you drag a point slot outside of the background image's margins, it will turn red. If you then release it, it will disappear and you will need to add it again.

Note: To remove a point, select the x below it.

Styling Points

Sizing a point

Use resize  to switch between three size options for the point slot:

- Point name and point value displayed next to each other (long)
- Point name displayed above point value (more compact)
- Point value only (smallest)

Customizing a point's color and opacity

1. Select the palette .
2. Do one of the following to choose a color:

- Use the color slider and move the selection circle.
 - Enter the **HEX** color code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
3. Do one of the following to change the opacity:
 - Use the opacity slider.
 - Change the seventh and eighth digits of the **HEX** code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
 4. Select an area outside of the palette to close it.

Adding Status Colors

When status colors are configured, a color-coded status bar displays on the left edge of the card's point slot. You can configure the status color to change depending on point's current value.

Using Premade Color Sets

1. Select Add colors  (on the point slot's left), which makes a window appear.
2. Choose a **Color Set** from the dropdown menu.
3. Enter the **Min value** and **Max value**.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

4. If you want this color configuration to apply to the text as well, select the **Apply color to text** checkbox.
5. Select **Save** to apply the status color configuration to the point.

Using a Custom Color Set

1. Select Add colors  (on the point slot's left), which makes a window appear.
2. From the **Color Set** dropdown menu, select **Custom**.
3. Enter the **Min value** and **Max value**.

Note: To add intermediate values, select the **+** (Add intermediate value). Then enter the new **Intermediate value**.

4. Select the thumbnails below the color spectrum, which opens a color palette.
5. Do one of the following to choose a color:

- Use the color slider and move the selection circle.
 - Enter the **HEX** color code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
6. Do one of the following to change the opacity:
 - Use the opacity slider.
 - Change the seventh and eighth digits of the **HEX** code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
 7. If you want this color configuration to apply to the text as well, select the **Apply color to text** checkbox.
 8. Select **Close**.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

9. Select **Save** to apply the status color configuration to the point.

Adding a Text Slot

1. Select **Add Point**.
2. Select **Text Slot**, which switches to a text editor tab.
3. Type and format text and/or hyper-linked text, as you would in a simple word processor.
4. Select **Save**.

Drawing and Configuring Canvas Zones

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

The procedures in this topic come *after* [Adding a Canvas on page 84](#) ([Accessing the Staging Area on page 84](#), [Uploading a Background Image on page 85](#), and [Adjusting the Margins on page 85](#)), but *before* [Naming the Canvas on page 85](#) and [Saving the Canvas on page 85](#).

Drawing a Zone and Assigning a Point

1. Select **Manage zones**.
2. Select **Draw zone**.

Note: After selecting **Draw zone**, a mouse's pointer will turn into a cross-hair once it moves across the margin and is over the background image. If the cross-hair moves off of the image before drawing a zone, it will revert to a pointer. Select **Draw zone** again.

3. Position the cross-hair onto a starting-point for the zone shape.
4. Single-click, which starts the drawing.
5. Move the cross-hair to the next point of the shape.

Note: A line follows the cross-hair from the previous point.

6. Single-click to make a side of the shape.
7. Repeat the previous two steps until all but the last side is made.
8. Double-click to finish the shape.

Note: The **Device** list and **Point Selector** immediately appear. The zone shape may seem to disappear in the background. However, it will still be there once you assign a point to it.

9. From the **Device** list and **Point Selector**, locate and choose the point that you want to assign to the zone.

Note: If creating on a global dashboard , a drop-down menu is above the **Device** list and **Point Selector**. If you want to select a point from a different project, select that project from the drop-down menu first.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

10. For the point slot (which is automatically added along with the zone shape), do *one* of the following:

- Drag the point slot to where you want it to display on the zone.

Note: In this case you should also style the point (see [Styling Points on page 86](#)) in relationship to the zone's colors (see [Configuring a Zone's Colors on page 90](#)).

- Hide the zone point (see [Hiding a Zone Point on page 90](#).)

11. When done drawing zones and/or configuring their colors (see [Configuring a Zone's Colors on page 90](#)), select **Save zones**.

Hiding and Showing Zone Points

Hiding a Zone Point

1. Select **Manage zones** (if not in zone management mode already).
2. Select the zone shape associated with the point slot that you want to hide.

Note: Small, blue diamonds and circles (which can be used to modify the zone shape) appear around the zone when it's selected.

3. Select  (hide zone point).

Showing a Zone Point

1. Select **Manage zones** (if not in zone management mode already).
2. Select the zone shape associated with the point slot that you want to show.

Note: Small, blue diamonds and circles (which can be used to modify the zone shape) appear around the zone when it's selected.

3. Select  (show zone point).

Configuring a Zone's Colors

Using Premade Color Sets

1. Select **Manage zones** (if not in zone management mode already).
2. Select the zone that you want to configure colors for.

Note: Small, blue diamonds and circles (which can be used to modify the zone's shape) appear around the zone when it's selected.

3. Select **Change zone colors** .
4. Choose a **Color Set** from the dropdown menu.
5. Enter the **Min value** and **Max value**.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

Note: Selecting **Custom** for **Color Set** at this point will make color thumbnails appear next to the values again. Select a thumbnail to make a color palette appear, allowing you to modify the preset color. (For details, see the relevant information in [Using a Custom Color Set on page 91.](#))

Note: To add intermediate values, with **Custom** selected for **Color Set**, select the + (**Add intermediate value**). Enter the new **Intermediate value** and select a color for it.

6. Select **Save** to apply the color configuration to the zone.
7. When done drawing zones and/or configuring their colors, select **Save zones**.

Using a Custom Color Set

1. Select **Manage zones** (if not in zone management mode already).
2. Select the zone that you want to configure colors for.

Note: Small, blue diamonds and circles (which can be used to modify the zone's shape) appear around the zone when it's selected.

3. Select **Change zone colors** .
4. Select **Custom** (the default) for the **Color Set** (if not selected already).
5. Enter the **Min value** and **Max value**.

Note: To add intermediate values, select the + (**Add intermediate value**). Then enter the new **Intermediate value**.

6. Select the gray thumbnails next to the values, which opens a color palette.
7. Do one of the following to choose a color:
 - Use the color slider and move the selection circle.
 - Enter the **HEX** color code.
 - Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.
8. Do one of the following to change the opacity:
 - Use the opacity slider.
 - Change the seventh and eighth digits of the **HEX** code.

- Select a previously used color and opacity setting from the rectangular swatches at the bottom of the palette.

9. Select an area outside of the palette to close it.

Note: See the preview of the color spectrum that will be applied to the entered range of values.

10. Select **Save** to apply the color configuration to the zone.

11. When done drawing zones and/or configuring their colors, select **Save zones**.

Adding a Report Module

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Prerequisites

Configure report settings in **Reports** . (In [Managing Reports on page 124](#), see the configuration topics for the needed report types.)

Adding the Report Module

1. Go to **Dashboards** , then select the dashboard that you want to display the report module on.

Note: Select a non-global dashboard. Currently, global  dashboards cannot display report modules.

2. Select **Add Instance**.

3. Select **Report**, which opens a report module staging area.

4. From the **Select Report** dropdown list, choose the report setting.

Note: The report setting's title (as configured in **Reports** ) auto-fills in **Report module title**. Enter a custom title if you want a different title to display on the dashboard.

5. Select **Add**.

6. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Note: *Report modules* span the entire width of a dashboard. Alternatively, create a *report card*, which has several sizes. (See [Creating a Report Card on page 71](#).) However, unlike a report module, once created, a report card cannot switch between report settings.

Modifying Report Modules

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Changing the Displayed Report

After [Adding the Report Module on page 92](#), you can change the report displayed by that dashboard report module. To change the displayed report:

1. From the dashboard report module that you want to change, select the pencil icon .
2. From the **Select Report** dropdown list, choose the report that you want the module to display.

Note: The displayed report changes, but the title of the dashboard report module stays the same.

3. Select the title of the dashboard report module, which opens an **Edit Report Title** window.
4. Modify the **Report Title** as necessary.
5. Select **Submit**.

Managing Schedules and Events

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Know Before Beginning

A KMC Commander **schedule** controls (with an optional default value) commandable points (for example, Active Cooling Setpoint) on one or more devices on a network. Particular times for the schedule to operate are controlled by **events**. An **event** controls start/stop time, priority, repetition, and the zones affected by the schedule actions.

To be able to command a point from a schedule (or card), the point must be selected in the **Points of Interest** column on the **Assign profile to [device name]** page. See [Assigning Device Profiles on page 43](#).

KMC Commander schedules write to a default priority of 16. If a KMC Commander schedule will command a device, change the **Schedule Write Priority** and **Holiday Schedule Write Priority** in **Settings > Protocols** to an appropriate (higher) level for the controlled devices (see [Configuring Protocol Settings on page 13](#)). To eliminate conflict with other schedules in a device, delete, set to NULL, or set to a lower priority level any schedules internal to the device or written by another device.

Inputs cannot be controlled directly. Map an input (for example, from an STE-6020 dial setpoint on AI2 or a relay contact on B3) into an AV or BV and control the mapped value.

To change a *current* setpoint, command the *active* heating/cooling setpoint rather than the (scheduled) occupied or unoccupied heating/cooling setpoint.

Adding Schedules

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For an explanation of the difference between **schedules** and **events**, as well as important details to know, see [Know Before Beginning on page 94](#) in [Managing Schedules and Events on page 94](#).

Adding Schedules (Simplified Method)

Using **Simplified**, you can select a **Type** of commandable point (for example, Active Cooling Setpoint) and any **device(s)** from the list which have that type of point. You can then apply the same **Active Value** for that point type to all selected devices.

1. Go to **Schedule Manager** , then **Configurations**.
2. Select **Add Schedule** to open the **Create Schedule** window.
3. Select **Simplified** (if not already selected).
4. Select (from the dropdown menu) the point **Type** that you want to command with the schedule.

Note: Once a **Type** is selected, devices that have that point type turn white. Devices that don't remain blue.

5. Select the orange checkboxes next to any devices with that point type that you want to apply the active value to.

Note: Devices that have that point type (white) can be selected. Devices that don't (blue) cannot.

Note: If you want to apply the active value to that point type in all (or most) devices that have it, select **Select All Available**. (You can then uncheck the boxes next to any devices that are exceptions.)

6. Enter the **Active Value**.

Note: The **Active Value** is the what KMC Commander will set the point to *after an event starts* (the schedule is **On**). (See [Creating Events on page 96](#).)

7. For **Reapply Schedule**:

- Choose **Enabled** if you want this schedule to reapply to the selected device(s) every 5 minutes during an event time-frame.

Note: This can restore (within 5 minutes) scheduled values lost during a device power outage or reboot.



Caution: Take care that two schedules do not command the same point at the same priority at the same time (due to overlapping events) with **Reapply Schedule** set to **Enabled**. If one schedule has it enabled, the values will bounce once between the two schedules. If both schedules have it enabled, the values will bounce continuously between the two for the duration of the overlapping time-frame.

- Leave it **Disabled** if losing the scheduled value due to complications is not a concern.

8. Enter a **Name** for the schedule.

9. Select **Save**.

Continue by [Creating Events on page 96](#).

Adding Schedules (Advanced Method)

Using **Advanced**, you can select individual points in individual devices. You can then apply unique **Active** and **Default** values to each of the selected points.

1. Go to **Schedule Manager** , then **Configurations**.
2. Select **Add Schedule** to open the **Create Schedule** window.
3. Select **Advanced**.
4. Below **Select device(s)**, select a device with points that you want to include in the schedule.

Note: When selected, a sublist of commandable points expands below the device.

5. Select the checkboxes next to any of the device's points that you want to command with the schedule.
6. Enter the **ON** and/or **OFF** values for each point.

Note: The **ON** value is what KMC Commander will set the point to *after an event starts* (the schedule is on). The **OFF** value is what it will set the point to *after an event ends* (the schedule is off). (See [Creating Events on page 96](#).) **Default** should generally be left blank (NULL), which returns the value after the end of an event to the value it was before the start of the event.

7. Repeat steps 4 to 6 for all devices and points that you want to include in the schedule.
8. For **Reapply Schedule**:
 - Choose **Enabled** if you want this schedule to reapply to the selected device(s) every 5 minutes during an event time-frame.

Note: This can restore (within 5 minutes) scheduled values lost during a device power outage or reboot.



Caution: Take care that two schedules do not command the same point at the same priority at the same time (due to overlapping events) with **Reapply Schedule** set to **Enabled**. If one schedule has it enabled, the values will bounce once between the two schedules. If both schedules have it enabled, the values will bounce continuously between the two for the duration of the overlapping time-frame.

- Leave it **Disabled** if losing the scheduled value due to complications is not a concern.

9. Enter a **Name** for the schedule.
10. Select **Save**.

Continue by [Creating Events on page 96](#).

Creating Events

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For an explanation of the difference between **schedules** and **events**, as well as important details to know, see [Know Before Beginning on page 94](#) in [Managing Schedules and Events on page 94](#).

1. Go to **Schedule Manager** , then **Calendar** (if not already selected).
2. On the calendar, select and drag from the start time to the stop time of the event you want to create, which opens the **Create Event** window.
3. Enter an **Event Name**.
4. Select the **Schedule** to use for this event from the drop-down list.

5. If necessary, adjust the event's **Start Time** and **End Time**.

Note: Select the date and time icon  to go to the date and time selector.

6. Choose a **Priority** for the event from the dropdown menu.

Note: KMC Commander schedules write to a default priority of 16. If a KMC Commander schedule will command a device, change the [Schedule Write Priority on page 16](#) and [Holiday Schedule Write Priority on page 16](#) in **Settings > Protocols** to an appropriate (higher) level for the controlled devices. To eliminate conflict with other schedules in a device, delete, set to NULL, or set to a lower priority level any schedules internal to the device or written by another device.

7. For weekly events, select **Weekly** from the **Repeat** dropdown menu, then select the checkboxes for the days of the week you want the event to repeat on.
8. Select **Save**.
9. Select **Close**, or repeat steps 3 through 8 to create more events using the window.

Modifying and Deleting Schedules

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Modifying a Schedule

1. Go to **Schedule Manager** , then **Configurations**.
2. In the row for the schedule that you want to change, select **Edit** .
3. Make any changes.

Note: See [Adding Schedules on page 94](#) for details.

4. Select **Update**.

Deleting a Schedule

1. Go to **Schedule Manager** , then **Configurations**.
2. In the row for the schedule that you want to remove, select **Delete** .
3. Select **Confirm**.

Modifying and Deleting Events

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Modifying an Event

Using the Update Event window

1. Go to **Schedule Manager** , then **Calendar** (if not already selected).
2. Select the event's tile on the calendar, which opens the **Update Event** window.
3. Make any changes.
4. Select **Update**.

Using its calendar tile

Note: You can work with an event's calendar tile to change its **Start Time** and **Stop Time**. For changing other properties, see [Using the Update Event window on page 98](#).

1. Go to **Schedule Manager** , then **Calendar** (if not already selected).
2. If you need to change the start time, drag the event across the calendar to the new start time.
3. If you need to change the duration, grab the bottom of the event and resize it to the new duration.

Deleting an Event

1. Go to **Schedule Manager** , then **Calendar** (if not already selected).
2. Select the event's tile on the calendar, which opens the **Update Event** window.
3. Select **Delete**.

Managing a BACnet Schedule Object

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

KMC Commander can read and write to a BACnet Schedule object's Out of Service and Present Value properties, as well as read it's Status Flags. Currently this is done separately from **Schedule Manager** .

Prerequisites

The procedures in this topic assume that the following have been done for the device of interest:

- [Configuring Networks on page 37](#)
- [Discovering Devices on page 43](#)
- [Assigning Device Profiles on page 43](#)

Also, the device's BACnet Schedule object Present Value data type configuration must be known. Check the device's documentation, and/or the device itself (using another tool).

Adding Schedule Object Properties to the Device Profile

Access the device's profile and discover points

1. Go to **Networks Explorer** , then **Networks**.
2. Select **View**  (in the row of the network that has the device with the Schedule object that you want to add).
3. Select **Edit Profile** (in the row of device with the Schedule object that you want to add).
4. Select **Discover** (to ensure that the schedule object points are discovered).

Add the Out of Service property

1. Locate the point named **SCH_### OOS** (for example, SCH_01 OOS).

Note: OOS stands for Out of Service.

2. Select **Select Type** in that row.
3. In the **Select Point Type** search field, begin typing "binary command".
4. Select **Binary Command**.

Note: Selecting the point type automatically checks the box in the **Points of Interest** column. To also trend the point, select the checkbox in the **Trend (his)** column.

Repeat for any other Schedule object Out of Service properties needed (for example, SCH_02 OOS).

Add the Status Flag property

1. Locate the point named **SCH_### SF**. (For example, SCH_01 SF.)

Note: SF stands for Status Flag.

2. Select **Select Type** in that row.
3. In the **Select Point Type** search field, begin typing "String".
4. Select **String**.

Note: Selecting the point type automatically checks the box in the **Points of Interest** column. To also trend the point, select the checkbox in the **Trend (his)** column.

Repeat for any other Schedule object Status Flag properties needed. (For example, SCH_02 SF.)

Add the Present Value property

1. Locate the point named **SCH_###**. (For example, SCH_01.)

Note: This is the Schedule object's Present Value property.

2. Select **Select Type** in that row.
3. Select (using the dropdown menu, search, or tree selector) a point type compatible with the device's BACnet schedule configuration. (See the [Prerequisites on page 98](#).)

Note: Selecting the point type automatically checks the box in the **Points of Interest** column. To also trend the point, select the checkbox in the **Trend (his)** column.

Repeat for any other Schedule object Present Value properties needed. (For example, SCH_02.)

Update the device profile

1. Select **Update Profile & Assign**.

Note: A list of all the devices that use this profile appears in an **Assign profile** window.

2. Select the checkboxes next to the devices that you want to assign this edited profile to.
3. Select **Assign to Devices**.

Note: **Regenerating Points** will appear at the bottom and will revert to **Assign Profile** when the process completes. It is okay to leave the page during the process. In the network's device list, a **spinning gear icon** will appear under **Actions** until the device profile has regenerated.

Creating a Custom Card for the Schedule Object

After [Adding Schedule Object Properties to the Device Profile on page 99](#):

Access the Custom card staging area

1. Go to **Dashboards**  and select the dashboard you want to add the card to.
2. Select **Add Instance**.
3. Select **Card**, which opens the card staging area. (**Custom card** is selected by default.)

Add the Out of Service property to a point slot

1. Select a point slot (**Select Point**), which makes the **Device** list and **Point Selector** appear.
2. From the **Device** list, locate and select the device.
3. In the **Point Selector**, locate **SCH_### OOS** (for example, SCH_01 OOS).
4. Select **SCH_### OOS**.

Add the State Flag property to a point slot

1. Select a point slot (**Select Point**), which makes the **Device** list and **Point Selector** appear.
2. From the **Device** list, locate and select the device.
3. In the **Point Selector**, locate **SCH_### SF** (for example, SCH_01 SF).

4. Select **SCH_### SF**.

Add the Present Value property to a point slot

1. Select a point slot (**Select Point**), which makes the **Device** list and **Point Selector** appear.
2. From the **Device** list, locate and select the device.
3. In the **Point Selector**, locate and select **SCH_###** (for example, SCH_01).
4. Select **SCH_###**.

Title and add the card

1. Enter a **Card title**.
2. Select **Add**.
3. Choose either **Add to Top of Dashboard** or **Add to Bottom of Dashboard**.

Writing to the Schedule Object's Present Value

After [Creating a Custom Card for the Schedule Object](#) on page 100:

Set the Schedule object to Out of Service

1. Select the **SCH_### OOS** point slot on the card.
2. Select **true**.
3. Select **Write**.

Note: To confirm Out of Service, refresh the dashboard until **SCH_### SF** reports **OutOfService**. It may take several minutes.

Write to the Present Value

1. Select **the SCH_###** point slot on the card.
2. Select the value that you want to write.
3. Select **Write**.

Managing Trends

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

About Trends

Trends show historical point data that can be tracked, saved, and exported. Points sharing a graph can be of different types (Analog, Binary, or Multi-State).

The interval at which trend points are read can be changed from the default (15 minutes). For example, the interval might be made longer to reduce data consumption through cellular connections or shorter to increase the update rate for critical environmental monitoring. See [Point Update Wait Interval \(Minutes\) on page 15](#) in **Settings**  **>Protocols**.

Creating Trend Graphs

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Creating a Trend Line Graph

[Access the Trend Graph Creation Area](#)

Go to **Trends** , then select **Add New Trend**.

Add Points

1. Select **Add Point**, which opens the **Add points to** __ window.
2. Locate and select the point that you want to display trend data from.

Note: Selecting a device from the **Device** list (left) narrows the **Point Selector** list (right) to show only the points in that device.

Note: You can filter both lists by typing in **Search Devices**. You can also filter the **Point Selector** list by typing in **Search Points**.

Note: Below a device name, the information in gray text is the device type, as set in the device's profile (see [Editing a Device Profile on page 45](#)). Below a point name, the information in gray text is [parent device name]:[point ID].

Note: As devices and points are filtered, the number of displayed devices or points out of the total (matching that criteria) is given at the bottom of each list.

Note: To display more devices or points in a list, select **Load More Devices** or **Load More Points** (at the bottom of each list).

3. Repeat step 2 for as many points as you want to add.
4. Select **Add**.
5. Select **Save**.

Set a View Mode

1. From the option buttons in the upper-right corner, choose the view mode (**Day, Week, Month, Year, ...**) that you want to become the **Default** for this trend graph.

Note: Select the ... (meaning more) to reveal a drop-down menu of more options.

2. Select **Save**.

Set a Binning (Averaging) Interval

Binning reduces noise by grouping raw data points into sets (bins) based on a time interval, averaging the mean value of each data bin, then presenting the averages (mean values) on the graph. Binning takes into consideration all data points, but reduces the amount of data points presented to help the viewer make a higher-level interpretation.

See [Use Cases on page 104](#) for examples of effective uses of binning.

To set the binning interval for the trend graph:

1. Choose an interval from the **Binning Interval** drop-down menu that is suitable for what needs to be presented.

Note: **Raw Data** means no binning interval will be applied. The graph will present all data points.

2. Select **Save**.

Set a Chart Type

1. From the **Chart Type** drop-down menu, select one of these options:
 - **Line** displays a line graph only.
 - **Line With Range** displays a shaded minimum and maximum value range behind a line graph.
 - **Range** displays a minimum and maximum value range only.

Note: **Line With Range** and **Range** only apply to binned data. (See [Set a Binning \(Averaging\) Interval on page 103](#).)

2. Select **Save**.

Name the Trend Graph

1. Replace "New Trend Graph" with a name for the trend graph.
2. Select **Save**.

Using the Binning Interval

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Benefits of Binning Data

Binning reduces noise by grouping raw data points into sets (bins) based on a time interval, averaging the mean value of each data bin, then presenting the averages (mean values) on the graph. Binning takes into consideration all data points, but reduces the amount of data points presented to help the viewer make a higher-level interpretation.

Use Cases

More Clearly Showing Trends for CO₂ and Humidity

Figure 1 shows **Raw Data** for CO₂ and humidity during October 2024. The line graphs based on the raw data show all the modulations throughout each day. A viewer must interpret the line graphs to understand the trends.

Figure 1:

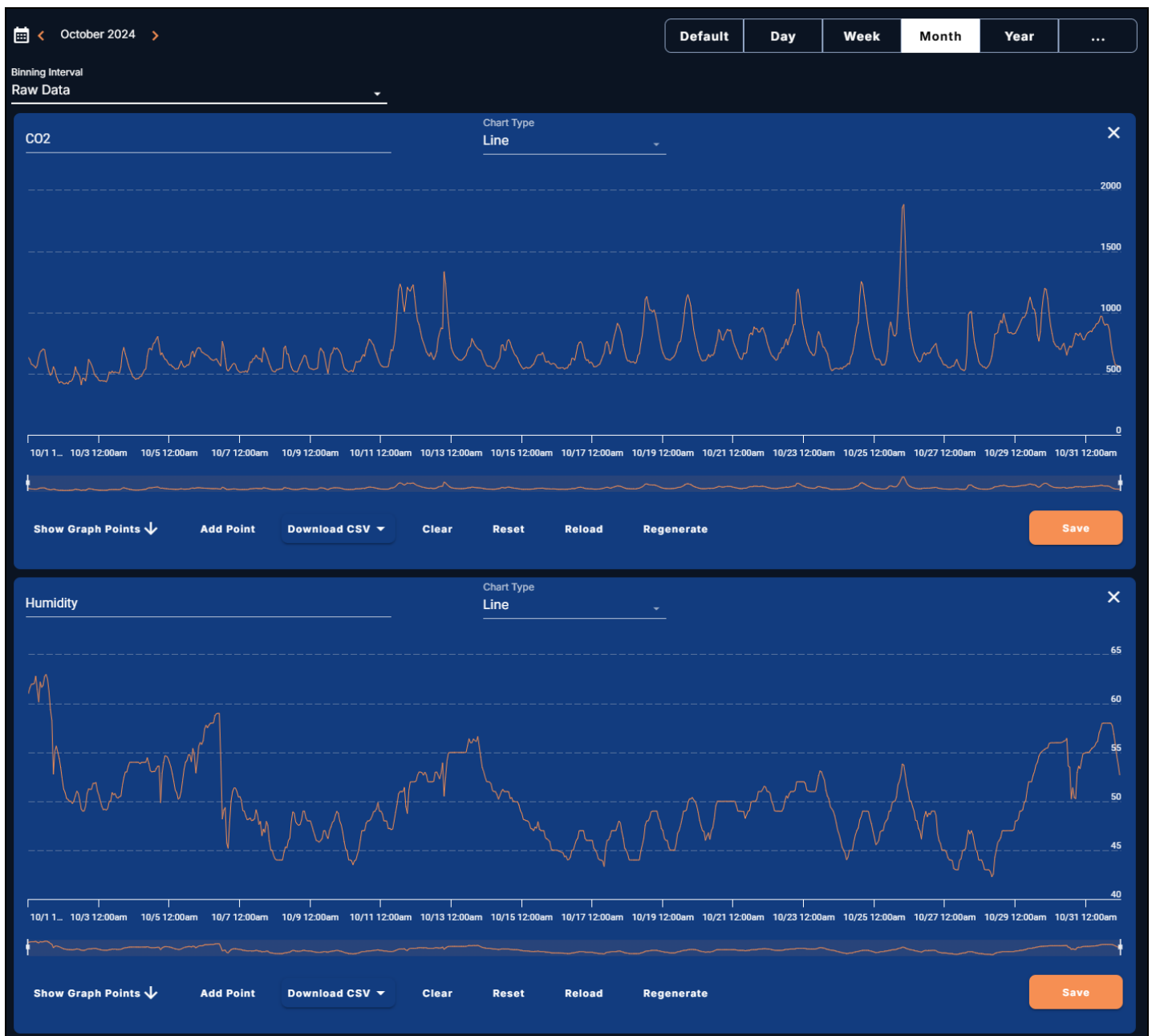
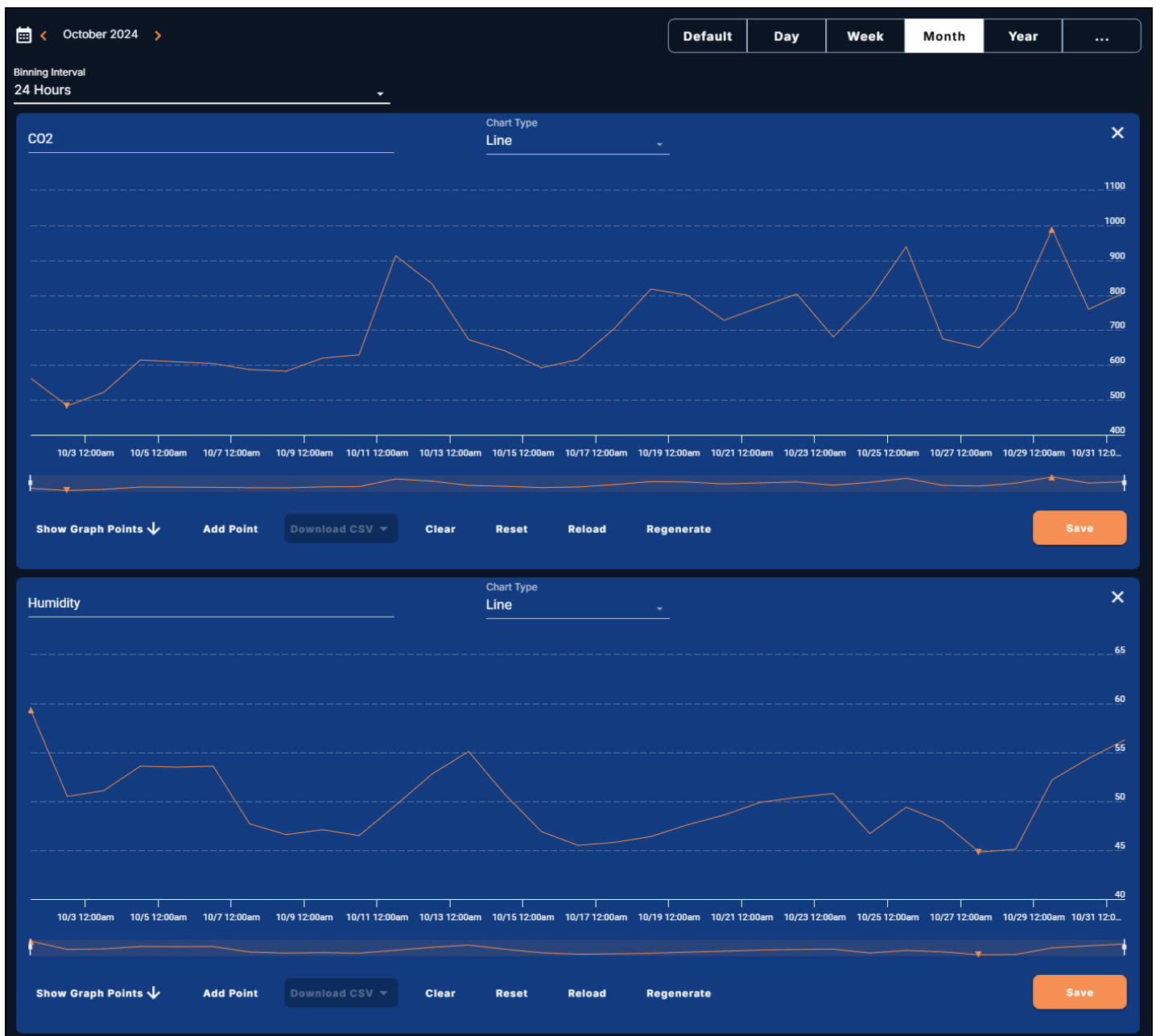


Figure 2 shows the data averaged at a **Binning Interval** of **24 Hours**. The line graphs now present more pronounced and clear trends. A viewer can more clearly see:

- an overall increase of CO₂ during October
- an overall decrease in humidity from October 1 to October 29
- an increase in humidity in the last few days of the month
- significant peaks in both CO₂ and humidity occurring around October 11–14

Figure 2:

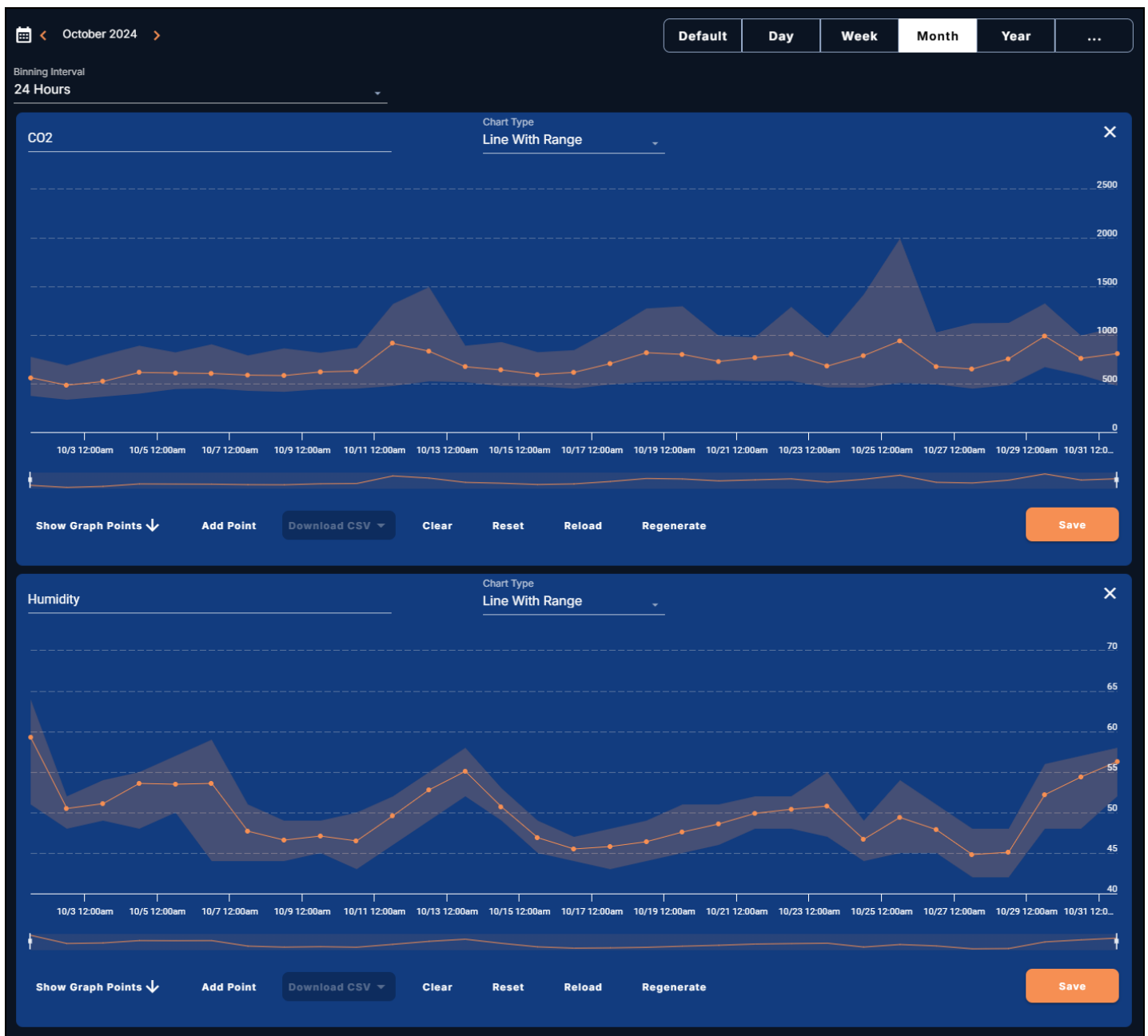


When the **Binning Interval** is set and the **Chart Type** is set to **Line**, the viewer can no longer see the amount of CO₂ and humidity modulation each day. Setting the **Chart Type** to **Line With Range** provides this perspective—while keeping the focus on the daily averages—by displaying a shaded minimum and maximum value range behind the line (see Figure 3).

Figure 3 more clearly shows, based on the averages, that:

- CO₂ fluctuates more than humidity
- humidity is more tightly controlled than CO₂

Figure 3:



From this binned **Line With Range** presentation of the data, facility management could determine to more tightly control CO₂ to improve the indoor air quality. If a customer was concerned about humidity, the data would demonstrate that on average the space is staying within 40–60% with controlled modulation.

Viewing Trends

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Viewing Information About a Trend

Go to **Trends** , then select the *chevron*  next to the name of a saved trend on the left pane.

Information expands below the trend's name about:

- the saved (default) **View Mode** (see [Choosing a View Mode on page 108](#))
- the saved **Binning Interval** (see [Choosing a View Mode on page 108](#))
- the saved **Chart Type** (see [Choosing a Chart Type on page 108](#))
- the saved **Points** (see [Add Points on page 102](#) in [Creating Trend Graphs on page 102](#))

Note: Selecting the *chevron* ▼ next to the trend's name will expand the trend's information only. Selecting the trend's *name* will also open the trend's graph on the right pane.

Viewing a Trend Graph

Accessing the Trend Graph

Go to **Trends**, then select the *name* of a saved trend on the left pane, which opens the trend graph in the right pane.

Note: Selecting the *chevron* ▼ next to the name expands the information about the trend only.

Choosing a View Mode

1. From the option buttons in the upper-right corner, chose the timeframe of data that you want to view.

Note: Select the ... (meaning more) to reveal a drop-down menu of more options.

Note: **Default** is the timeframe that was saved when creating the trend graph. (See [Set a View Mode on page 103](#) in [Creating Trend Graphs on page 102](#).)

2. Use the < (left arrow) and > (right arrow) to move the timeframe backward and forward through the available data history.

Choosing a Binning (Averaging) Interval

Binning reduces noise by grouping raw data points into sets (bins) based on a time interval, averaging the mean value of each data bin, then presenting the averages (mean values) on the graph. Binning takes into consideration all data points, but reduces the amount of data points presented to help the viewer make a higher-level interpretation.

See [Use Cases on page 104](#) for examples of effective uses of binning.

Choose an interval from the **Binning Interval** drop-down menu that is suitable for what you want to see presented.

Note: **Raw Data** means no binning interval will be applied. The graph will present all data points.

Choosing a Chart Type

From the **Chart Type** drop-down menu, select one of these options:

- **Line:** displays a line graph only

- **Line With Range**: displays a shaded minimum and maximum value range behind a line graph
- **Range**: displays a minimum and maximum value range only

Note: **Line With Range** and **Range** only apply to binned data. (See [Choosing a Binning \(Averaging\) Interval on page 108.](#))

Viewing Information About a Precise Point

To view information about a precise point on the graph, hover over that point, which makes a box with detailed information appear.

Zooming In on a Trend Graph

To zoom in on a section of a trend graph, adjust the sliders at the bottom of the graph.

Note: If you want multiple open trend graphs to zoom to the same timeframe, select **Zoom Sync**. When you adjust the sliders of one trend graph, the sliders of all trend graphs will adjust accordingly.

Note: Select **Default** to go back to the original timeframe, or choose an alternate timeframe. (See [Choosing a View Mode on page 108.](#))

Refreshing a Trend Graph

A loaded trend in **Trends**  or **Dashboards**  does not automatically refresh its data. After sufficient time has passed for an update, try one or more of the following actions:

- Select **Reload** from the trend graph window.
- Select a view mode.
- Select the back  and forward  arrows (at the top next to the date of the timeframe).
- Reselect the trend from the list of saved trends on the left pane.
- Refresh your browser.

Hiding and Showing Trend Lines

1. With a trend graph open on the right pane in **Trends**  (see [Viewing a Trend Graph on page 108](#)), select **Show Graph Points**.
2. Select the **Eye** icon in the point slot matching the color of the trend line that you want to hide/show.

Modifying and Deleting Trend Graphs

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Adding a Point to a Trend Graph

1. Go to **Trends** , then select the *name* of the trend from the list on the left pane.
2. Select **Add Point**, which opens the **Add a points to ___** window.
3. Select a device on the left pane.
4. Select a point in that device on the right pane.
5. Select **Add**.
6. Select **Save**.

Deleting a Point from a Trend Graph

1. Go to **Trends** , then select the *name* of the trend from the list on the left pane.
2. Select **Show Graph Points**.
3. Select the **x** beside the point that you want to delete from the trend graph.
4. Select **Save**.

Hiding and Showing Trend Lines

1. With a trend graph open on the right pane in **Trends**  (see [Viewing a Trend Graph on page 108](#)), select **Show Graph Points**.
2. Select the **Eye** icon in the point slot matching the color of the trend line that you want to hide/show.

Deleting a Trend

1. Go to **Trends** , then select the trend from the list on the left pane.
2. Select **Delete**  below the expanded information about the trend.
3. Select **Confirm**.

Exporting Trend Data

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Exporting Trend Data as a CSV File

To export a trend's data as a CSV file:

1. Go to **Trends** , then select the trend from the list on the left pane.
2. Select **Download CSV** (on the right pane next to **Add Point**).
3. From the dropdown options, select either **Current View** (exports data for all visible points) or select an individual point's data to export.

Note: If the CSV file has no data, the selected point was hidden by selecting the **x** beside it.

Managing Alarms

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Know Before Beginning

Before any new alarm will trigger for a condition, its target point must return into a normal condition for at least one minute (which resets the alarm).

If an alarm is configured for a target point that is already in an alarm condition during configuration, no alarm will trigger until after the target point has gone into the normal condition first. For example, if an alarm is configured for a duct temperature, but the wiring to the duct temperature sensor is defective, no alarm will trigger for the abnormal duct temperature until after the defective wiring is fixed.

The interval that alarm points are read can be changed from the default. For example, the interval might be made longer to reduce network traffic or shorter to increase the update rate for critical environmental monitoring. (See both the [Individual Point Intervals on page 13](#) and the [Point Update Wait Interval \(Minutes\) on page 15](#) in **Settings**  **> Protocols**.)

Configuring a Point Value Alarm

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Access Point Value alarm configuration

1. Go to **Alarms** , then **Configurations**.
2. Select **Add Configuration**, which opens the **Add Alarm Configuration** window.
3. From the **Category** drop-down menu, select **Point Value**.

Select the Target Network, Target Device, and Target Point

1. Choose the **Target Network** from the drop-down list.

Note: While choosing, you can type the first letter of the network's name to jump closer to its location in the list.

2. Choose the **Target Device** from the drop-down list.

Note: While choosing, you can type the first letter of the device's name to jump closer to its location in the list.

3. Choose the **Target Point** from the drop-down list:

- If you selected a **binary** or **multistate** target point, continue to [For a binary or multistate target point on page 113](#).
- If you selected an **analog** target point, continue to [For an analog target point on page 114](#).

Note: While choosing, you can type the first letter of the point's name to jump closer to its location in the list.

For a binary or multistate target point

After [Configuring a Point Value Alarm on page 112](#), select an alarm **Type** from the dropdown menu:

- If you selected **Exact Value**, continue to [With an Exact Value type alarm on page 113](#).
- If you selected **Dynamic**, continue to [With a Dynamic type alarm on page 113](#).

Note: A dynamic alarm compares the values of two related points in one or two devices.

With an Exact Value type alarm

1. Choose the **Target Value** (for example, OFF).
2. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the value must be the target value (for example, OFF) before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\) on page 15](#).

3. Continue to [Finish Configuring the Alarm on page 115](#).

With a Dynamic type alarm

1. Choose the **Dynamic Reference Device** (the same or another device) from the dropdown list.
2. Choose the **Dynamic Reference Point** (that will be compared to the **Target Point**) from the dropdown list.
3. From the **Dynamic Type** dropdown menu, choose either **Equal To** or **Not Equal To**.
4. From the Choose a **Match Method** (**State Text** or **Numeric Value**)

Note: The **State Text** match method compares the state text tags (for example: inactive/ active, OFF/ ON) that were given to the numeric values of the target and dynamic reference points that you chose. The **Numeric Value** match method compares the numeric values of the points.

Note: KMC Commander shows the **Target Point State Texts** and **Dynamic Point State Texts** for the points that you chose. If the state texts of the two points are exactly the same, you could choose either **State Text** or **Numeric Value**. However, if the state texts are *not* exactly the same, selecting **Numeric Value** can prevent the alarm from triggering improperly.

5. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the values must be equal or not equal before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\)](#) on page 15.

6. Continue to [Finish Configuring the Alarm](#) on page 115.

For an analog target point

After [Configuring a Point Value Alarm](#) on page 112, select an alarm **Type** from the dropdown menu:

- If you selected **Range**, continue to [With a Range type alarm](#) on page 114.

Note: A range alarm compares the value of the target point to a defined range of acceptable values.

- If you selected **Dynamic**, continue to [With a Dynamic type alarm](#) on page 114.

Note: A dynamic alarm compares the values of two related points in one or two devices.

With a Range type alarm

1. Enter the **Target Value**.
2. Choose a **Range Type** from the dropdown menu.
3. Enter the **Range Value** (for example, 5° lower than the target value).
4. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which, for example, the value must be 5° lower than the target value before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\)](#) on page 15.

5. Continue to [Finish Configuring the Alarm](#) on page 115.

With a Dynamic type alarm

1. Choose the **Dynamic Reference Device** (the same or another device) from the dropdown list.

2. Choose the **Dynamic Reference Point** (that will be compared to the **Target Point**) from the dropdown list.
3. Choose the **Range Type** from the dropdown menu.
4. Enter the **Range Value** (for example, the dynamic reference point is 5° higher than the target point).
5. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which, for example, the dynamic reference point must be 5° higher than the target point before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\)](#) on page 15.

6. Continue to [Finish Configuring the Alarm](#) on page 115.

Finish Configuring the Alarm

1. After [Configuring a Point Value Alarm](#) on page 112, choose the alarm's level of **Priority (Info, Warning, or Critical)**.
2. If needed, choose a **Notification Group** from the dropdown list.

Note: Choosing a notification group is optional (such as for **Info** level alarms), but is recommended for **Critical** and **Warning** level alarms.

Note: To send alarm messages, first create a notification group (with email addresses and/or cell phone numbers) in **Settings**  > **Users, Roles, & Groups** > **Groups**. (See [Configuring \(Alarm Notification\) Groups](#) on page 25.)

3. Enter an **Alarm Name**.
4. Choose whether to make **Acknowledgment** of the alarm **Required** or **Not required**:
 - **Required**—The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be orange. Hovering over the button makes an "Ack. Required" tip appear.
 - **Not Required**—The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be black. Hovering over the button makes an "Ack. Optional" tip appear.
5. Optionally, enter a **Message**, which will appear in **Alarm Manager** and optional emails and/or text notifications.
6. If you want to duplicate this alarm to similar devices or points, select a **Duplication Strategy**.

Note: See [Duplicating Alarms](#) on page 120 for more information.

7. Select **Add**.

Configuring a Gateway Status Alarm

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure a Gateway Status Alarm

1. Go to **Alarms** , then **Configurations**.
2. Select **Add Configuration**, which opens the **Add Alarm Configuration** window.
3. From the **Category** drop-down menu, select **Gateway Status**.
4. From the **License** drop-down menu, select the license associated with the gateway that you want to configure the status alarm for.

Note: Go to **Settings**  > **Project** > [License Information on page 12](#) to find the name of the license associated with the gateway of the project you are currently in.

Note: Licenses are filtered based on your SI company and the current project. If you are logged into a project that doesn't belong to your SI company, you will not see any licenses in the drop-down menu.

5. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the gateway must go offline (or come back online) before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\) on page 15](#).

6. Choose the alarm's level of **Priority** (**Info**, **Warning**, or **Critical**).
7. If needed, choose a **Notification Group** from the dropdown list.

Note: Choosing a notification group is optional (such as for **Info** level alarms), but is recommended for **Critical** and **Warning** level alarms.

Note: To send alarm messages, first create a notification group (with email addresses and/or cell phone numbers) in **Settings**  > **Users, Roles, & Groups** > **Groups**. (See [Configuring \(Alarm Notification\) Groups on page 25](#).)

8. Enter an **Alarm Name**.
9. Choose whether to make **Acknowledgment** of the alarm **Required** or **Not required**:
 - **Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be orange. Hovering over the button will make an "Ack. Required" tip appear.

- **Not Required**—The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be black. Hovering over the button will make an "Ack. Optional" tip appear.

10. Optionally, enter a **Message**, which will appear in **Alarm Manager** and optional emails and/or text notifications.
11. Select **Add**.

Configuring a Network Status Alarm

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure a Network Status Alarm

1. Go to **Alarms** , then **Configurations**.
2. Select **Add Configuration**, which opens the **Add Alarm Configuration** window.
3. From the **Category** drop-down menu, select **Network Status**.
4. Choose the **Target Network** from the drop-down list.

Note: While choosing, you can type the first letter of the network's name to jump closer to its location in the list.

5. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the network must go offline (or come back online) before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\) on page 15](#).

6. Choose the alarm's level of **Priority (Info, Warning, or Critical)**.
7. If needed, choose a **Notification Group** from the dropdown list.

Note: Choosing a notification group is optional (such as for **Info** level alarms), but is recommended for **Critical** and **Warning** level alarms.

Note: To send alarm messages, first create a notification group (with email addresses and/or cell phone numbers) in **Settings**  > **Users, Roles, & Groups** > **Groups**. (See [Configuring \(Alarm Notification\) Groups on page 25](#).)

8. Enter an **Alarm Name**.
9. Choose whether to make **Acknowledgment** of the alarm **Required** or **Not required**:

- **Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be orange. Hovering over the button will make an "Ack. Required" tip appear.
- **Not Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be black. Hovering over the button will make an "Ack. Optional" tip appear.

- Optionally, enter a **Message**, which will appear in **Alarm Manager** and optional emails and/or text notifications.
- Select **Add**.

Configuring a Device Status Alarm

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure a Device Status Alarm

- Go to **Alarms** , then **Configurations**.
- Select **Add Configuration**, which opens the **Add Alarm Configuration** window.
- From the **Category** drop-down menu, select **Device Status**.
- Choose the **Target Network** from the drop-down list.

Note: While choosing, you can type the first letter of the network's name to jump closer to its location in the list.

- Choose the **Target Device** from the drop-down list.

Note: While choosing, you can type the first letter of the device's name to jump closer to its location in the list.

- Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the device must go offline (or come back online) before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\)](#) on page 15.

- Choose the alarm's level of **Priority (Info, Warning, or Critical)**.
- If needed, choose a **Notification Group** from the dropdown list.

Note: Choosing a notification group is optional (such as for **Info** level alarms), but is recommended for **Critical** and **Warning** level alarms.

Note: To send alarm messages, first create a notification group (with email addresses and/or cell phone numbers) in **Settings**  > **Users, Roles, & Groups** > **Groups**. (See [Configuring \(Alarm Notification\) Groups on page 25.](#))

9. Enter an **Alarm Name**.
10. Choose whether to make **Acknowledgment** of the alarm **Required** or **Not required**:
 - **Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be orange. Hovering over the button will make an "Ack. Required" tip appear.
 - **Not Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be black. Hovering over the button will make an "Ack. Optional" tip appear.
11. Optionally, enter a **Message**, which will appear in **Alarm Manager** and optional emails and/or text notifications.
12. If you want to duplicate this alarm to similar devices or points, select a **Duplication Strategy**.

Note: See [Duplicating Alarms on page 120](#) for more information.

13. Select **Add**.

Configuring a Point Status Alarm

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Configure a Point Status Alarm

1. Go to **Alarms** , then **Configurations**.
2. Select **Add Configuration**, which opens the **Add Alarm Configuration** window.
3. From the **Category** drop-down menu, select **Point Status**.
4. Choose the **Target Network** from the drop-down list.

Note: While choosing, you can type the first letter of the network's name to jump closer to its location in the list.

5. Choose the **Target Device** from the drop-down list.

Note: While choosing, you can type the first letter of the device's name to jump closer to its location in the list.

6. Choose the **Target Point** from the drop-down list.

Note: While choosing, you can type the first letter of the point's name to jump closer to its location in the list.

7. Enter the **Threshold (seconds)**.

Note: Threshold is the number of seconds for which the point must go offline (or come back online) before KMC Commander triggers the alarm.

Note: Threshold is also affected by the polling rate set in **Settings**  > **Protocols**. See [Point Update Wait Interval \(Minutes\)](#) on page 15.

8. Choose the alarm's level of **Priority (Info, Warning, or Critical)**.

9. If needed, choose a **Notification Group** from the dropdown list.

Note: Choosing a notification group is optional (such as for **Info** level alarms), but is recommended for **Critical** and **Warning** level alarms.

Note: To send alarm messages, first create a notification group (with email addresses and/or cell phone numbers) in **Settings**  > **Users, Roles, & Groups** > **Groups**. (See [Configuring \(Alarm Notification\) Groups](#) on page 25.)

10. Enter an **Alarm Name**.

11. Choose whether to make **Acknowledgment** of the alarm **Required** or **Not required**:

- **Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be orange. Hovering over the button will make an "Ack. Required" tip appear.
- **Not Required** —The **Acknowledge all** button for this alarm (when triggered in **Alarm Manager**  > **List**) will be black. Hovering over the button will make an "Ack. Optional" tip appear.

12. Optionally, enter a **Message**, which will appear in **Alarm Manager** and optional emails and/or text notifications.

13. If you want to duplicate this alarm to similar devices or points, select a **Duplication Strategy**.

Note: See [Duplicating Alarms](#) on page 120 for more information.

14. Select **Add**.

Duplicating Alarms

Note: An alarm can only be duplicated during its initial setup.

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Duplicating an Alarm to Similar Devices

1. Configure all of the alarm's properties, as described in [Configuring a Point Value Alarm](#) on page 112.

2. For **Duplication Strategy**, select **Device profiles**.
3. Select the dropdown box to see a list of all devices that have the same device profile as the alarm's **Target Device**.

Note: Only devices that have been discovered and assigned profiles in **Networks Explorer**  can appear in the list. For information on discovering devices and assigning profiles to them, see [Discovering Devices on page 43](#) and [Assigning Device Profiles on page 43](#).

4. Select the checkboxes for devices that you want to duplicate the alarm to.
5. Select **Add**.

Duplicating an Alarm to Similar Points

1. Configure all of the alarm's properties, as described in [Configuring a Point Value Alarm on page 112](#).
2. For **Duplication Strategy**, select **Point Tags**.
3. Under **Point search on**, select **Name**, **Type**, and/or **Instance** to see other points in the list with the same name, type, or instance as the alarm's **Target Point**.
4. Select the checkboxes for points that you want to duplicate the alarm to.
5. Select **Add**.

Modifying and Deleting Alarms

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Modifying an Alarm

1. Go to **Alarm Manager** , then **Configurations**.
2. Select **Edit**  in the row of the alarm that you want to modify, which opens the **Edit Alarm Configuration** window.
3. Make any changes.

Note: See [Configuring a Point Value Alarm on page 112](#) for details.

4. Select **Update**.

Deleting an Alarm

1. Go to **Alarm Manager** , then **Configurations**.
2. In the row for the alarm that you want to remove, select **Delete** .
3. Select **Confirm**.

Finding, Viewing, and Acknowledging Alarms

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Video Overview

Finding Alarms

1. Go to **Alarm Manager** , then **List** (if not already selected).
2. Use any of the following methods to find alarms of interest:
 - Choose whether to view the alarms by **Day**, **Week**, or **Month**.
 - Use the forward arrow  and back arrow  to view the next or previous day, week, or month.
 - From the **Show** dropdown, choose whether to show (within the currently displayed time-frame) **All**, **Active only**, **Non-acknowledged only**, **Acknowledged only**, **Critical only**, **Warning only**, **Info only**, or **Archived only** alarms.

Note: Auto-archiving of alarms is configured in **Settings**  > **Project** > [Auto Archive Alarms on page 9](#).

- From the **Device** dropdown
- Enter a term into the search  to only see alarms containing that term (within the currently displayed time-frame).

Viewing Alarm Details

Alarm Bell and Color	Priority Level	Alarm State
Green bell ringing 	Info	Currently in alarm condition
Green bell with line 	Info	Not currently in alarm condition
Yellow bell ringing 	Warning	Currently in alarm condition
Yellow bell with line 	Warning	Not currently in alarm condition
Red bell ringing 	Critical	Currently in alarm condition
Red bell with line 	Critical	Not currently in alarm condition

After [Finding Alarms on page 122](#), select the row for an alarm, which expands the alarm's details below the row.

If an alarm has triggered multiple times within the selected time-frame, the number of instances will display next to the alarm bell. The multiple instances will be grouped under a single, folded row. After expanding the row, scroll down to see all the instances.

The **Target Device**, **Target Point**, **Priority**, and **Alarm Group** (Users Notified) are configured when setting up the alarm. (See [Configuring a Point Value Alarm on page 112](#).)

Alarmed Value is what the **Target Point**'s value was when it triggered the alarm.

When a user acknowledges an alarm (see [Acknowledging Alarms on page 123](#)), **Ack. by [User's Name]** appears under **Acknowledged**. If the user entered an acknowledgment message, view it by hovering over the message balloon , or see the **Message** in the expanded alarm details. **Acknowledged at** displays the date and time when the user acknowledged the alarm.

Acknowledging Alarms

Acknowledging Alarms by Folded Row

After [Viewing Alarm Details on page 122](#) and taking any necessary action:

1. Select **Acknowledge all** in the folded row for that alarm, which opens the **Alarm Acknowledgment** window.

Note: An orange button means that acknowledgment is required. A black button means that acknowledgment is not required. (See [Configuring a Point Value Alarm on page 112](#).)

2. Enter an acknowledgment **Message**, or leave it blank ("N/A" will display as the message).
3. Select **Continue**.

Note: Your **Message** replaces the default alarm message, and **Ack. by [Your Name]** appears under **Acknowledged**.

Acknowledging Multiple Folded Rows

After [Viewing Alarm Details on page 122](#) and taking any necessary action:

1. Select the checkboxes for the folded rows that you want to acknowledge.
2. Select **Acknowledge** (orange button above the rows).
3. Enter an acknowledgment **Message**, or leave it blank ("N/A" will display as the message).
4. Select **Continue**.

Note: Your **Message** replaces the default alarm messages, and **Ack. by [Your Name]** appears under **Acknowledged**.

Acknowledging Alarm Instances Individually

After [Viewing Alarm Details on page 122](#) and taking any necessary action:

1. Select the folded alarm row, which expands the individually triggered instances below.
2. From an individual alarm instance, select **Acknowledge**.
3. Enter an acknowledgment **Message**, or leave it blank ("N/A" will display as the message).
4. Select **Continue**.

Managing Reports

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Reports Overview

Using **Reports** , you can configure parameters for reports of the following types:

- **All Points:** Presents all points of interest in a project as a list.
- **All Points By Device:** Presents points of interest in a grid view. Device names are the rows and point names are the columns.
- **Energy Usage (Continuous Accumulators)**
- **Energy Usage (Hourly Accumulators)**
- **Energy Usage (From Power)**
- **Manual Override**
- **Out of Range**
- **Runtime**

You can use the parameters to view report results once, or name and save the parameters as a report *setting* to be run again later. A snapshot of the results of a report setting can be downloaded as a CSV file.

Once a report setting is saved, you can display the report on a dashboard using either a **Report Card** or **Report** module. A **Report Card** displays a single report in several sizes. (See [Creating a Report Card on page 71](#).) A **Report** module always spans the entire width of a dashboard, but easily switches between report settings. (See [Adding a Report Module on page 92](#)).

Important to Know

Each time you create or open a report setting the report runs and displays the available data *from that moment*. See the notes in the following sections of this guide for details about how to see subsequent changes to point values.

When running a saved report setting, if checkboxes next to particular points were selected before the initial save the report will only include results from those points. See the note in [Running a Saved Report Setting on page 134](#) for how to determine which points are included. Add a new report to search all points of interest in the project again.

When running an **Energy Usage** report, for accurate results you must choose the right type (**Continuous Accumulators**, **Hourly Accumulators**, or **From Power**) for the device and/or situation. See [Choosing the Right Report Type on page 128](#) for guidance.

The results of a **Manual Override** report setting are based on the last time KMC Commander read the priority. If KMC Commander was used to make all overrides, then the report results will be up-to-date when you run the setting. However, if overrides may have been made by other means you will need to make KMC Commander read the priority again to ensure up-to-date results. See the notes in [Configuring a Manual Override Report on page 130](#) for how to check and update the **Priority Last Read** time.

Configuring an All Points Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, select **All Points**.

Note: A table appears showing the values (at the **Value Last Read** time) of all licensed points of interest in the project.

Note: Switch **Show Time** to **On** to see a table column showing the **Value Last Read** time.

Note: To see any changes to point values after the **Value Last Read** time, first wait for the duration of the **Point Update Wait Interval**, then close the new report's window and repeat steps 1 and 2.

Note: The **Point Update Wait Interval** is in **Settings**  > **Protocols**. (See [Point Update Wait Interval \(Minutes\) on page 15](#) in [Configuring Protocol Settings on page 13](#) for details.)

3. Do any of the following to locate and view particular points:
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.

4. Take any desired **Actions** on points by selecting **Set Value/Clear Override**.

Note: This opens a window where basic and advanced settings can be adjusted for the point. For details, see [Using a Report on page 136](#).

5. If you want a snapshot of these report results, select **Download CSV**.
6. **Close** the new report's window.

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in steps 1 to 5 of [For viewing once on page 125](#).
2. Check the boxes next to any points that you will later want to run the report on again.

Note: If you don't choose any points, the saved report settings will include all the points of interest in the project.

3. In the text field at the top of the new report's window, clear **New Report** and enter a descriptive name.
4. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

5. **Close** the saved report's window.

Note: If you want to create and save multiple reports from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list on the left pane.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Configuring an All Points By Device Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, select **All Points By Device**.
3. Select **Search Devices**, which makes an **Add devices to New Report** window appear.
4. Check the boxes next to any devices that you want to include in the report.

Note: These devices will later become the rows of the report.

Note: Type in **Search Devices** to narrow the list.

Note: If you want to include all devices, you can either use **Select All** in this window, or select **All Devices** in the report configuration window.

5. Select **Add**.

Note: The names of the devices appear under **Selected Devices**.

6. Select the field under **Select Points**, which causes a list of every unique point name from the **Selected Devices** to appear.
7. Check the boxes next to any point names that you want to include in the report.

Note: Each unique point name becomes a column of the report. Avoid many similar columns by naming point types consistently across devices. (See [Assigning Device Profiles on page 43](#) and [Editing a Device Profile on page 45](#).)

Note: To check the boxes next to all the point names, use **Select All**.

8. Select an area outside of the list to close the list and see the report.

Note: **Search Points** is an alternative to first using **Search Devices** then using the **Select Points** list. Selecting **Search Points** causes a **Device List** and **Point Selector** to appear (like when adding points on a card). These allow you to select particular points from particular devices, and adds device and point names at once to the report. (For details on using the **Device List** and **Point Selector**, see the related information in [Changing or Adding Points on a Card on page 73](#).)

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in [For viewing once on page 126](#).
2. In the text field at the top of the report's window, clear **New Report** and enter a descriptive name.
3. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

4. **Close** the saved report's window.

Note: If you want to create and save multiple reports from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list on the left pane.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Configuring an Energy Usage Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

Choosing the Right Report Type

A device that you can run an energy usage report for likely has one or both of the following points:

- A point with **kW** or **W** units, associated with the device's power meter, which measures *the rate of energy transfer* at any given moment
- A point with **kWh** or **Wh** units, associated with the device's meter accumulator, which *totals the energy used* over a period of time (based on frequent readings of the power meter)

The **Energy Usage (From Power)** report takes the available readings of a point with **kW** or **W** units (a point associated with a device's power meter) and calculates the energy used over the chosen time period (a day, a week, or a month). The **Energy Usage (From Power)** report functions like an accumulator in this way. However, since KMC Commander reads the point only as frequently as the **Point Update Wait Interval** setting (every one minute to an hour), it is not as accurate as the *device's* accumulator, which takes readings every second or faster. It may be accurate enough in some cases.

The **Energy Usage (Continuous Accumulators)** and **Energy Usage (Hourly Accumulators)** reports find a point with **kWh** and **Wh** units (the point associated with a device's meter accumulator). Therefore, these report types are usually more accurate. However, you must know which type of accumulator the device has (continuous or hourly) and look at the matching report type to see the correct **Total Usage (kWh)** value. If you do not have that information at hand, look at a trend of the point:

If the trend looks like this...	...use this type of report:
	Energy Usage (Continuous Accumulators)
	Energy Usage (Hourly Accumulators)

The **Energy Usage (Continuous Accumulators)** report sums the increases of the point's kWh values over the chosen time period. (The report ignores decreases in the value caused by the accumulator resetting.) Continuous accumulators are the most common type of accumulator for power meters, so this report type is most commonly used. This report type also gives the most accurate **Total Usage (kWh)** result (if the device has a continuous accumulator).

The **Energy Usage (Hourly Accumulators)** report averages the readings of the point's hourly kWh values over the chosen time period and multiplies by the number of hours. Because the report is based on an average, the **Total Usage (kWh)** result is an approximation.

If trend data is not readily available and the device has a point with kWh or Wh units, run all three types of Energy Usage reports, then compare the **Total Usage (kWh)** results. The report with a **Total Usage (kWh)** result closest to that of the **Energy Usage (From Power)** report is most likely the correct report to run (matches that device's type of accumulator).

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, choose the right **Energy Usage** report type for your purpose.

Note: See [Choosing the Right Report Type on page 128](#) for guidance.

3. Make a selection from the **Choose time Period** dropdown menu.

Note: Switch **Show Time** to **On** if you want columns that show the **Start Time** and **End Time** of the energy usage (in addition to the **Start Date** and **End Date**).

4. Do any of the following to locate and view particular points:
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.
5. If you want a snapshot of these report results, select **Download CSV**.
6. **Close** the new report's window.

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in steps 1 to 5 of [For viewing once on page 129](#).
2. Check the boxes next to any points that you will later want to run the report on again.
3. In the text field at the top of the new report's window, clear **New Report** and enter a descriptive name.
4. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

5. **Close** the new report's window.

Note: If you want to create multiple report settings from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list on the left pane.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Configuring a Manual Override Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, select **Manual Override**.

Note: A table appears showing points at or below **Priority level 8** (based on their last known priority values at the **Priority Last Read** time), that have been overridden (prior to the **Value Last Read** time).

Note: Switch **Priority Last Read** to **On** to see a table column showing the **Priority Last Read** time.

Note: To update a point's **Priority Last Read** time, select **Update Priority Arrays**.

Note: Switch **Value Last Read** to **On** to see a table column showing the **Value Last Read** time.

Note: To see any changes to the **Current Value** after the **Value Last Read** time, first wait for the duration of the **Point Update Wait Interval**, then close the new report's window and repeat steps 1 and 2.

Note: The **Point Update Wait Interval** is in **Settings**  > **Protocols**. (See [Point Update Wait Interval \(Minutes\)](#) on page 15 in [Configuring Protocol Settings](#) on page 13 for details.)

3. Do any of the following to locate and view particular points:
 - Increase or decrease the **Priority level** at or below which points are included in the table.
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.
4. Take any desired **Actions** on points by selecting **Set Value/Clear Override** .

Note: This opens a window where basic and advanced settings can be adjusted for the point. For details, see [Using a Report](#) on page 136.

5. If you want a snapshot of these report results, select **Download CSV**.
6. **Close** the new report's window.

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in steps 1 to 5 of [For viewing once on page 130](#).
2. Check the boxes next to any points that you will later want to run the report on again.

Note: If you don't select any points, the saved report will still be able to search all points in the project.

3. In the text field at the top of the new report's window, clear **New Report** and enter a descriptive name.
4. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

5. **Close** the new report's window.

Note: If you want to create and save multiple reports from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Configuring an Out of Range Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, select **Out of Range**.

Note: A table appears showing points with values that fall outside (at the **Value Last Read** time) the default limits. To see any points that subsequently go out of range, close the new report's window and repeat steps 1 and 2.

Note: Switch **Show Time** to **On** to see a table column showing the **Value Last Read** time.

Note: To see any changes to a point value after the **Value Last Read** time, first wait for the duration of the **Point Update Wait Interval**, then close the new report's window and repeat steps 1 and 2.

Note: The **Point Update Wait Interval** is in **Settings**  > **Protocols**. (See [Point Update Wait Interval \(Minutes\)](#) on page 15 in [Configuring Protocol Settings](#) on page 13 for details.)

3. Do any of the following to locate and view particular points:
 - Increase or decrease the **point values above** and/or **point values below** which points are shown in the table.
 - Switch **On** or **Off** the upper or lower limit.
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.
4. Take any desired **Actions** on points by selecting **Set Value/Clear Override** .

Note: This opens a window where basic and advanced settings can be adjusted for the point. For details, see [Using a Report](#) on page 136.

5. If you want a snapshot of these report results, select **Download CSV**.
6. **Close** the new report's window.

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in steps 1 to 5 of [For viewing once](#) on page 131.
2. Check the boxes next to any points that you will later want to run the report on again.

Note: If you don't choose any points, the saved report will be able to search all points of interest in the project.

3. In the text field at the top of the new report's window, clear **New Report** and enter a descriptive name.
4. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

5. **Close** the new report's window.

Note: If you want to create and save multiple reports from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Configuring a Runtime Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About

For viewing once

1. Go to **Reports** , then select **Add New Report**.
2. From the **Choose a Report** dropdown menu, select **Runtime**.
3. Make a selection from the **Choose time Period** dropdown menu.

Note: The runtimes of any points of interest tagged binary will appear in the table.

4. Do any of the following to locate and view the run-times of particular points:
 - Switch **Show Time** to **On** to see columns that show the run **Start Time** and **End Time** (in addition to the **Start Date** and **End Date**).
 - Switch **Duty Cycle** to **On** if see a column that shows **Duty Cycle (%)**.
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.
5. If you want a snapshot of these report results, select **Download CSV**.
6. **Close** the new report's window.

To save to run again later

Note: When you save a report, you save that report *setting* to be run again when you later reopen the report. To save a snapshot of a report's *results*, select **Download CSV**.

1. Complete the actions described in steps 1 to 5 of [For viewing once on page 133](#).
2. Check the boxes next to any points that you will later want to run the report on again.
3. In the text field at the top of the new report's window, clear **New Report** and enter a descriptive name.
4. Select **Save**.

Note: The saved report's name, with details expanded below it, appears in the list on the left pane.

5. **Close** the new report's window.

Note: If you want to create and save multiple reports from this same open window, be sure to select **Save New** for each subsequent report. If you select **Save**, the new configuration will overwrite the old one in the **Reports** list.

To display the report on a dashboard, continue to [Adding a Report Module on page 92](#) and/or [Creating a Report Card on page 71](#).

Running a Saved Report Setting

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Run a Report Setting Command

1. Go to **Reports** , then select the name of the saved report setting (from the **Reports** list on the left pane).

Note: The saved report setting runs again at the moment that you open it. On **All Points** and **Manual Override** reports, the updated **Report time** displays near the top of the report's window. On **Energy Usage** and **Runtime** reports, the updated time is reflected by an updated **Start Date** and **End Date** in the table.

Note: The **Updated At** date and time (in the expanded details on the left pane) is the last time *the report setting was modified* (**Save** was selected). It is not the last time the report was run.

2. Do any of the following to locate and view particular saved points:
 - Filter the displayed points by typing in **Search on Point/Device**.
 - Increase or decrease the number of **Rows per page**.
 - Use the previous and next arrows to display points listed on other pages.

Note: The report runs again only on the points that were selected when the report setting was first saved. These are listed next to **Points** in the expanded details on the left pane. For saved **Manual Override** and **Out of Range** report settings, adjusting the priority level or max and min values will not return points outside of the set listed in the expanded details. To search all points of interest in the project again, create a new report by selecting **Add New Report**.

Note: If **Search All** is next to **Points** (in the expanded details on the left pane) all points of interest in the project were included when the report setting was initially saved.

3. For **All Points**, **All Points By Device**, **Manual Override**, and **Out of Range** reports, take any desired **Actions** on points by selecting **Set Value/Clear Override** .

Note: This opens a window where basic and advanced settings can be adjusted for the point. For details, see [Using a Report on page 136](#).

4. If you want a historical snapshot of these report results, select **Download CSV**.

Modifying and Deleting a Saved Report Setting

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Modifying a Saved Report Setting

After a report setting is saved, you can later modify and save the following:

- The name of the report
- Whether or not time columns show in the table by default
- The default priority level at or below which points will display for a **Manual Override** report
- The default min and max values for an **Out of Range** report

Note: The points included in a saved report setting are listed next to **Points** in the report setting's expanded details on the left pane. To add other points of interest from the project to a report setting (or remove points), create a new report by selecting **Add New Report**.

Note: If **Search All** is next to **Points** (in the expanded details on the left pane) all points of interest in the project were included when the report setting was initially saved.

To Modify a Saved Report Setting:

1. Go to **Reports** , then select the name of the saved report setting from the **Reports** list on the left pane.
2. Make any of the changes listed above that are necessary.
3. Select **Save**.

Note: The **Updated At** date and time (in the expanded details on the left pane) immediately changes to show the time of the save.

Deleting a Saved Report Setting

1. Go to **Reports** , then select the saved report setting (from the **Reports** list on the left pane).

Note: You can expand the report setting's details but avoid opening it by selecting the *chevron*  next to its name.

2. In the area with details, which expands below the name, select **Delete** .
3. Select **Confirm**.

Note: If the (now deleted) report setting was opened on the right pane, **Close** it.

Linking to a Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Link to a Report

Note: You can obtain the unique URL for a saved report setting, then link to it directly (for anyone with project access).

1. Go to **Reports** .
2. Select (from the **Reports** list on the left pane) the report setting that you want to link to.
3. Select **Link**  (below the report setting's expanded details), which copies the unique URL to your clipboard.
4. Share the link with other users who have project access.

Using a Report

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Writing to a Point

You can write to points from these report types: **All Points**, **All Points By Device**, **Manual Override**, and **Out of Range**.

Using the simplified method

1. From the report's **Actions** column, in the row of a writable point, select **Set Value/Clear Override** .

Note: A window opens with the device and point name.

2. Enter or select a new value.
3. Select **Write Priority [Default]**.

Note: The priority given here is the [Default Manual Write Priority on page 15](#), configured in **Settings**  **> Protocols**.

Note: The value will be written for the duration of the [Manual Write Timeout on page 15](#) (default **None**), configured in **Settings**  **> Protocols**.

Using Advanced Settings

1. From the report's **Actions** column, in the row of a writable point, select **Set Value/Clear Override** .
2. Enter or select a new value.
3. Select **Show Advanced Settings**, which expands to allow you to:
 - Select a **Write Priority** from the dropdown menu.
 - Select a **Write Timeout** from the dropdown menu.

Note: **Write** should be selected by default (for **Write Value** or **Clear Slot**).

Note: A history of the current and previous 10 reads of the priority array displays below. Scroll to the right to view all 10. The interval of the time stamps is partly determined by the [Read Priority Array Wait Interval \(Minutes\)](#) on page 15.

4. Select **Write Priority** _.

Note: It may take some time for the point on the *device* to change to the new value so that the report shows the change. See also the [Read Time After Point Writes \(Seconds\)](#) on page 10, configured in **Settings**  > **Protocols**.

Clearing a Priority

1. From the report's **Actions** column, in the row of a writable point, select **Set Value/Clear Override** .
2. Select **Show Advanced Settings**.
3. For **Write Value** or **Clear Slot**, select **Clear**.
4. From the **Clear Priority** dropdown menu, choose the priority that you want to clear.

Note: A history of the current and previous 10 reads of the priority array displays below. Scroll to the right to view all 10. The interval of the time stamps is partly determined by the [Read Priority Array Wait Interval \(Minutes\)](#) on page 15.

5. Select **Clear Priority** _.

Note: It may take some time for the point on the *device* to clear the value so that the report shows the change. See also the [Read Time After Point Writes \(Seconds\)](#) on page 10, configured in **Settings**  > **Protocols**.

Changing Personal Profile Settings

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Changing Your Personal Information

To change the first name, last name, email address, office phone number, or cell phone number stored in your profile:

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **Personal Info** (selected by default).
3. Make the desired changes.
4. Select **Save**.

Note: Here you can also switch between the old (before February 2022) UI and **New** (since February 2022) **UI**.

Configuring SMS Notifications

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **Personal Info** (selected by default).
3. As needed, enter, change, or leave as is your **Cell Phone** number.
4. Either switch on or off **Use Cell Phone for SMS Notifications** (as desired).
5. Select **Save**.

Note: **Admin** role users can configure other users' cellphone and SMS notification preferences if those users haven't previously set them up. (Go to **Settings**  > **Users/Roles/Groups** > **Users**, then select **Edit User** .)

Changing Your Password

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **Account**.
3. Enter your **Current Password**.
4. Enter a **New Password**.

Note: Passwords should be at least 8 characters long.

5. **Confirm New Password**.
6. Select **Save**.

Accessing Your API Key

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **API**.
3. Select **Copy to Clipboard**.

Note: See also [API Integration on page 146](#).

Configuring Your Offline Device Notifications

To receive (or stop receiving) email notifications for projects when controllers or other devices go offline:

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **Notifications**.
3. Check (or uncheck) the box next to any projects that you want (or don't want) to receive notifications for.
4. Select **Save**.

Switching Between Projects

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

When logged in, to switch to another project:

1. Select the circle with your initials, which makes a menu appear.
2. Select **Change Project**.

Note: The name of the project that you are currently on shows below the option.

3. **Select a project** from the dropdown list.

Note: A Google map with red pins can show the location of projects if the addresses are entered in the (Cloud) KMC license administration. (To use this feature, supply KMC Controls with your desired project address information for the license server.) Select a red pin, then **Click to Continue** to open that project.

Add-ons (and Data Explorer)

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

What are Add-ons?

Add-ons are extra features that may appear after clicking the Cloud icon (below the Dashboard icon). Unless additional add-ons are purchased, the Data Explorer is the only available add-on. See [Using Data Explorer on page 141](#).

Add-ons can provide a means to automate VAV zoning, provide real-time monitoring of air quality, provide additional advanced analytics of data, perform drag-and-drop visual programming, and provide a tool for troubleshooting.

Using Data Explorer

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)



Caution: Data Explorer is for advanced users! Deleting the wrong objects could make your project **UNUSABLE**! Do NOT DELETE ANYTHING (using the trash can icon at the left end of the row) found in the results unless you are very sure about it. After deletion, you cannot undo it. All changes are done instantly.

Note: Data Explorer appears as a (default) add-on for users with a role given access to Data Explorer. (See [Defining Roles on page 24](#) and [Adding and Configuring Users on page 18](#).)

Note: For tips on using Data Explorer, select the **info icon** found above the Data Query search field.

About Haystack Tags

Tags are automatically attached to points of interest that appear in profiles and cards. In a device profile, if **Point of Interest** is checked, a “pointOfInterest” tag is attached to that point. If “trend” is checked, a “his” tag (the Haystack standard for history) is attached to that point. (See [Assigning Device Profiles on page 43](#).)

The Data Explorer allows searching for Haystack tags in the database. Tags are name/value pairs associated with a point or property. Most tags (searched for in Data Explorer) are either a marker tag (generally what type of thing it is) or a “property” tag (some type of value of the thing).

The marker tag format is just the **marker**. Examples include:

- point
- air and temp

The general property tag format is **tag=value**. The value can be an ID or contiguous string. Strings with a space must be enclosed in double quotes. Examples include:

- curVal>=50
- dis=SPACE

- `dis="METER 1"`

To combine marker and property tags, the format is **property tag=value and marker tag**. Examples include:

- `dis=weeklySetpointChange` and `schedule`
- `dis="PRI ACTUAL FLOW"` and `bacnetCur=AV24`
- `priority=warning` and `alarm` (results would give only active alarms, not any that have been acknowledged)

Note: For general information about Haystack tags, see project-haystack.org/tag and project-haystack.org/doc/TagModel. See also the video [KMC 101: What is Project Haystack?](#)

Performing a Search

1. Enter the query/search phrase in **Data Query**.

Note: For more guidance about forming search phrases, see [Tips for Queries on page 142](#).

2. Select **Search**.
3. Use the scroll bars to view all the columns and rows. (These change according to the kind of results.)
4. Select the name of a column to sort the rows.

Note: For more details about sorting, see [Tips for Sorting on page 143](#).

Note: To optionally export and download search results as a CSV file, select **Download CSV**.

Note: A CSV file download will use the currently selected sort method for each page. The CSV file will contain all the records for the query term, but each page is sorted separately from other pages. Use the sort features in Excel to refine the order.

Tips for Queries

For best results, choose the most specific query string that returns concise but complete records.

Searches are case sensitive. If a search fails, check for capitalization issues.

Search supports **and**, **or**, **=** (equal), **!=** (not equal), **<**, **>**, **<=**, **>=**, and **phrases in** (straight, not smart) **quotes**.

Common search terms include: **temp**, **humidity**, **sensor**, **point**, **device**, **schedule**, **alarm**, **space**, **air**, **dashboard**, **customDeck**, **trendSetting**, **sp** (setpoints), **his** (histories or trends), **dis** (display names), and **curVal** (current values).

Note: Searches must pair a property tag with a value. Searching for just a property tag will not return results.

Valid types of searches would include these examples:

- `dis=REL_HUMIDITY`

- `dis="STE-6020 Temp. Sensor #10"`
- `type=spaceTemperature`
- `objectName=SPACE_SENSOR`
- `dis="ANALOG_VALUE"`
- `curStatus!=ok`
- `unit=°F`

Note: The degrees symbol ° can be created by pressing the Alt key while typing 0176 on the numeric keypad.

- `trendSetting` and `dis="Voltage Meter"`
- `message="Who Changed the Setpoint?!"`
- `occ=true`
- `curStatus=down`
- `deviceRef=06e1a461-209f-11e8-832a-f714c13eae1`
- `modelName="BAC-9311CE"`
- `ip="10.110.4.95"`

Examples of database items that are not “points” are device, network, schedule, profile, trendSetting, and customDeck.

Tips for Sorting

Clicking a sort arrow sorts only the *current page* of multiple pages (if any). A fixed number of records are returned for each page. Use the left/right arrows at the bottom right to see other pages.

A selected sort tag will continue to sort other pages that are loaded, such as using the arrow/page buttons or starting a new search. The chosen sort tag is active until a new sort tag is chosen or the Data Explorer page is refreshed.

Sorting can be performed on marker and property tags but not for columns of lists (objects).

Adding Objects and Tags

Objects and tags can be added in Data Explorer if desired. A tag is a name and value pair applied to an entity that defines a fact or attribute about the entity. In Data Explorer, objects and tags can be added for storing values in the database that are not part of a device, for easier searching in Data Explorer of “favorite” objects, and for interfacing with APIs.

To add a new object:

1. Select **Add New Object**.
2. Select the appropriate tag **Type** from the drop-down box.

Note: For tag type descriptions, see the table in [Tag Types on page 144](#).

3. Enter a **Tag** name (e.g., justDesserts) and (if not a Marker tag) a Value (e.g., Goopy Goodness and Justice).
4. Select **Submit**.

Note: This will create an object with one tag.

To add tags to the new (or an existing) object:

1. Find the desired object by searching in Data Explorer.
2. Select the + sign at the (left) end of its line (next to the trash can).
3. Select the appropriate tag **Type** from the drop-down box.

Note: For tag type descriptions, see the table in [Tag Types on page 144](#).

4. Enter a **Tag** name and (if not a Marker tag) a **Value**.
5. Select **Add**.
6. Search for the same object.
7. Verify that the tag was created as intended and add more tags as desired.

Tag Types

Tag Type	Description	(Example) Name*	(Example) Value
Marker	A description that has no separate, associated value (usually used to indicate classifications or types of identity)	candyBar	(N/A)
Bool	A Boolean expression that is “true” or “false”	dessertTime	true
Number	An integer or floating point number	servings	4.5
Str	A string of Unicode characters (commonly used for human-read-able descriptions)	flavor	Milk Chocolate
Ref	A reference to another entity (requires a specific subset of ASCII characters: a-z, A-Z, 0-9, underscore, colon, dash, dot, or tilde)	dessertRef	1234abc

Tag Type	Description	(Example) Name*	(Example) Value
Coord	A geographic coordinate in latitude/longitude formatted as Q(lat,lng)	newParis_Ind	Q(41.499998,-85.9833294)
*Tag names must: • Contain only ASCII letters, digits, or underscore (a-z, A-Z, 0-9, _). • Start with an ASCII lower-case letter (a-z). • Use camel case (e.g., yummyCandyBar) by convention.			

Node-RED Add-on

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

What is Node-RED?

Node-RED is a drag-and-drop visual programming tool with a browser-based editor, which provides advanced users a means of expanding and integrating KMC Commander's capabilities. For more information, see the *KMC Commander and Node-RED Application Guide*. (See [Accessing Other Documents on page 164.](#))

API Integration

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Go to api.kmccommander.com for KMC Commander's API documentation.

Accessing Your API Key

1. Select the **circle with your initials**, which makes a menu appear.
2. Go to **Profile**, then **API**.
3. Select **Copy to Clipboard**.

Information

API (Application Programming Interface) integration provides a means of expanding the capabilities of KMC Commander for advanced users. See the video [KMC 101: What is an API?](#)

KMC Commander has a RESTful web service, in adherence with Representational State Transfer (REST), which is an architectural style that defines a specific set of constraints to be used for creating web services and provides interoperability between computer systems on the Internet. RESTful principles are client/server, stateless, cacheable, layered, and with a uniform interface. See restfulapi.net for more information about REST.

For information about Haystack tags, see:

- project-haystack.org
- the video [KMC 101: What is Project Haystack?](#)

Troubleshooting

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Note: For a list of useful network and installation troubleshooting tools, see [Software Tools on page 159](#).

“500 Internal Error” on Login Screen

- This may appear during a system update, boot, or reboot. Allow the gateway to fully restart, which can take at least several minutes, and refresh the screen.

Add-on Is Not Available

- If the Add-Ons (cloud) icon only shows Data Explorer, you do not have the needed extra license for Node-RED. Contact KMC Controls for additional purchase. See [Add-ons \(and Data Explorer\) on page 141](#).

Alarms Do Not Work

- Check the alarm configuration. See [Managing Alarms on page 112](#).
- Before any new alarm will trigger for a condition, its target point must return into a normal condition for at least one minute (which resets the alarm).

Communication Problems (BACnet MS/TP)

Note: BACnet MS/TP communication requires a BACnet router (e.g., [BAC-5051E](#)) connected between the MS/TP network and the Ethernet port on the gateway.

Note: For BACnet IP communication, see also [Communication Problems \(Internet/Ethernet\) on page 148](#).

Note: Multiple networks may be needed to see all the desired devices.

Note: Entering a known range of device instances will speed up the discovery process, but if devices are not found as expected, try expanding the range or select **Any**.

- Check that the gateway has adequate power. See [Power Problems on page 157](#).
- Review [Configuring a BACnet Network on page 37](#).
- If devices are local, do not enter a router IP address.
- Verify that port and BACnet network numbers match what your devices and BACnet routers are expecting.
- Verify that all desired devices are online.
- Check that the Ethernet ports are active. See [Configuring LAN/Ethernet Settings on page 28](#).

- Restart the gateway. See [Restarting the Gateway on page 162](#).
- Restart the BACnet router connected to the gateway.
- Check the wiring. See [Connecting Ethernet \[Advantech\]](#) or [Connecting Ethernet \[Dell\]](#), and [Connecting Power \[Advantech\]](#) or [Connecting Power \[Dell\]](#).
- Verify that all devices are set for the same the baud rate.
- Check the MAC address and Max Master of every connected device.
- Systematically isolate parts of the network to track down the problem.
- Check with the building's IT department that they are not blocking BACnet traffic.
- If available, use the capture and diagnostics in a [BAC-5051E BACnet router](#) to facilitate troubleshooting.
- Use Wireshark (see [Software Tools on page 159](#)) or equivalent software to find problems in network communication.
- Verify that the BACnet router has the latest available firmware.
- Take steps to minimize unnecessary traffic on the network, such as verifying that Control Basic is correctly managing the interval for reading from and writing to off-panel objects with WAIT, ALIAS, or NETPOINT functions.

Communication Problems (Cellular)



Caution: Do not accidentally disable a port needed for communication! If all ports are disabled, all communication with the KMC Commander IoT gateway is lost! The gateway would then need to be reset to the factory defaults. (See [Reset/Restore to Factory Defaults on page 158](#)).

- Check that the gateway has adequate power. See [Power Problems on page 157](#).
- Enable Cellular communication. See [Configuring Cellular Settings on page 30](#).
- Verify that the SIM card is inserted properly into the slot. See [Installing Optional Cellular](#).
- Go to **Settings > Ethernet 1 [eth0] / Ethernet 2 [eth1]**. Ensure that **Isolate IPv4 to Local Subnet** and/or **Isolate IPv6 to Local Subnet** is set to **Yes**. See [Configuring LAN/Ethernet Settings on page 28](#).
- Contact the cellular carrier.

Note: Verizon may take an hour after registration to begin proper cellular communication.

Communication Problems (Internet/Ethernet)



Caution: Do not accidentally disable a port needed for communication! If all ports are disabled, all communication with the KMC Commander IoT gateway is lost! The gateway would then need to be reset to the factory defaults. (See [Reset/Restore to Factory Defaults on page 158](#)).

Note: All network and Internet communication issues in a building should be resolved before installing KMC Commander!

- See [Ethernet Connection Indicators Are Unlit](#) on page 153.
- Check the Ethernet wiring. See [Connecting Ethernet](#) [Advantech] or [Connecting Ethernet](#) [Dell].
- Check the Ethernet network.
- Check that the gateway has adequate power. See [Power Problems](#) on page 157.
- Only one Ethernet port (e.g., 1 or 2, but not both) should have a live Internet connection. The two Ethernet ports should not have the same IP address.
- Use the correct password. See the KMC Commander Quick Connection Guide (see [Support Documents and Resources](#)).
- During initial setup, check that the (Internet) network connection has a DHCP server to get an address, and the PC being used is set to have a dynamic IP address rather than a static address.
- Verify that proper ports are being used.
- Check that necessary ports and addresses have been whitelisted. See [Configuring Whitelist/Blacklist Settings](#) on page 33.
- Check with the IT department that outbound TCP/IP ports 80, 123, and 443 are open. See the KMC Commander Network Information Technical Bulletin. (see [Support Documents and Resources](#)).
- Restart the gateway. See [Restarting the Gateway](#) on page 162.
- Enter the correct IP address. Check with the building's IT department and verify the correct Ethernet information is entered under Settings. See [Configuring LAN/Ethernet Settings](#) on page 28.
- If the IP address of either port on the KMC Commander IoT gateway has a conflict with the IP address of another device, the gateway will typically stop communicating on that port. Resolve the conflict and restart the gateway. See [Restarting the Gateway](#) on page 162.
- If the IP address was changed and then forgotten (and Wi-Fi is disabled), use Wireshark or other IP scanning tool on a different PC to sniff the network while restarting the gateway or contact KMC Controls for assistance. See [Restarting the Gateway](#) on page 162 and [Contact Information](#) on page 166.
- See [Communication Problems \(BACnet MS/TP\)](#) on page 147.
- See also ["Your Connection Was Interrupted" Error](#) on page 161.
- Temporarily bypass a switch or router to isolate any problems.
- Use routers and/or switches on the network that meet all the latest standards.
- See also [VPN Problems](#) on page 160.

Communication Problems (Wi-Fi)



Caution: Do not accidentally disable a port needed for communication! If all ports are disabled, all communication with the KMC Commander IoT gateway is lost! The gateway would then need to be reset to the factory defaults and all stored data would be deleted. (See [Reset/Restore to Factory Defaults](#) on page 158).

Note: A KMC Commander IoT gateway, with an active dynamic Internet connection on Port 1, starts transmitting its SSID about three minutes after its power is connected during initial setup.

Note: A live Internet connection through Ethernet Port 1 must be present for Wi-Fi to work during setup. During initial setup, Wi-Fi is for laptop communication with the gateway. The wired Ethernet Internet connection is for Cloud communication and updates. The (Internet) network connection must have a DHCP (Dynamic Host Configuration Protocol) server to get an address, and the PC being used for setup must be set to have a dynamic IP address rather than a static address.

Note: See also [Communication Problems \(Internet/Ethernet\)](#) on page 148.

- Check that the gateway has adequate power. See [Power Problems](#) on page 157.
- Verify that the gateway has been powered for at least three minutes (Power and Cloud LEDs are glowing solid green). See [Restarting the Gateway](#) on page 162.
- Verify that an Ethernet cable with an active dynamic Internet connection is plugged into Port 1.
- Go to the correct IP address in the browser. See [Connecting Wi-Fi and Making Initial Login](#).

Note: For initial setup, type 192.168.4.1 into the address field.

- Use the correct password. See the KMC Commander Quick Connection Guide (see [Support Documents and Resources](#)).
- Position the PC no more than about five feet away from the gateway for adequate signal strength. If the signal is weak, move the PC closer to the gateway.
- If the gateway is mounted inside a metal enclosure, leave the enclosure's door open during setup. If the door must remain closed, mount the antenna remotely. See [\(Optionally\) Installing the Antennas Remotely](#).
- Check that Wi-Fi is turned on in the PC and operating correctly.
- Look for the correct **CommanderBX-[service tag number on your individual KMC Commander IoT gateway]** SSID (for example, **CommanderBX-GG22B02**). See [Connecting Wi-Fi and Making Initial Login](#).
- Allow the connection even though the KMC Commander's certificate is "not trusted." See ["Your Connection Is Not Private" Error](#) on page 161.
- In the Windows **Network and Sharing Center**, check that the connection is open, and under **Set Network Location**, select **Public network**. See [Connecting Wi-Fi and Making Initial Login](#).

- Restart the gateway. See [Restarting the Gateway on page 162](#).
- Restart the PC.
- If the gateway can be accessed through Ethernet, go to **Settings** and check that **Wi-Fi** is enabled and set to **Access Point** rather than **Client**. See [Connecting Wi-Fi and Making Initial Login](#).

Note: An alternate method of accessing and setting up an gateway (instead of using Wi-Fi) is by connecting a laptop to Port 2 (Dell) / LAN A (Advantech) with an Ethernet patch cable and opening a browser window with address 192.168.2.1. This method still requires a live Internet connection to Port 1 / LAN A, and it may require temporarily changing the IP address on your computer. (For detailed instructions, see the [KMC Commander Port 2 Ethernet Connection Application Guide](#).)

Controller Operation Override Problems

- If used, writable points in cards are written to a default priority of 8, and schedules are written to a default priority of 16. Verify this is compatible with the controller programming and priority level defaults. If not, select a different priority. See [Schedules \(or Events\) Do Not Work on page 158](#), [Using Cards on page 75](#), and [Configuring Protocol Settings on page 13](#).
- See the Control Basic help system.

Decks (or Cards within Decks) Are Not Visible

Note: The Dashboard allows the cards to be organized into collections of “decks” (groups of cards) for easier navigation. Multiple Dashboards can be created and selected.

- Ensure the correct Dashboard is selected. See [Selecting a Dashboard to View on page 56](#).
- Repeat the steps to add cards and/or decks. See [Creating and Adding Cards on page 59](#) and [Creating and Adding Decks on page 79](#).
- See [Discovered Devices and Points Are Not Visible on page 152](#).

Device Discovery Problems

- Wait a few minutes and try again.

Note: Heavy network traffic will slow discovery and even cause timeouts.

Note: When KMC Commander transmits a device ID range for discovery of devices, it listens for responses. Other device activity on the network may return extraneous results from multiple devices outside that range.

- See [Discovering Devices on page 43](#).
- See [Communication Problems \(BACnet MS/TP\) on page 147](#).

- See [Communication Problems \(Internet/Ethernet\)](#) on page 148
- See [Communication Problems \(Wi-Fi\)](#) on page 150.
- Use Wireshark to check the network for devices.

Devices Down/Offline Today Email Notification

- See [Communication Problems \(Internet/Ethernet\)](#) on page 148 or [Communication Problems \(BACnet MS/TP\)](#) on page 147.

Devices Stopped Communicating

- Check that the device is not offline.
- Check the device configuration.
- See [Communication Problems \(Internet/Ethernet\)](#) on page 148 or [Communication Problems \(BACnet MS/TP\)](#) on page 147.

Discovery Problems

Note: Multiple networks may be needed to see all the desired devices.

Note: Entering a known range of device instances will speed up the discovery process, but if devices are not found as expected, try expanding the range or select **Any**.

- If discovery ends in a timeout, enter a smaller or known range of device instances.
- For BACnet MS/TP networks, see [Communication Problems \(BACnet MS/TP\)](#) on page 147.
- For BACnet IP/Ethernet networks see [Communication Problems \(Internet/Ethernet\)](#) on page 148.

Discovered Devices and Points Are Not Visible

- Ensure the devices were configured properly. See [Configuring Devices](#) on page 43.
- Ensure the devices are checked (not unchecked) in the site topology. See [Configuring a User's Topology Access](#) on page 18.

Drag-and-Drop in Site Explorer Does Not Work

- Drag the point or device until the green check mark (instead of red NO symbol) shows. See [Creating a Site Topology](#) on page 47.
- See [Points or Devices Not Visible in Site Explorer](#) on page 155.

Drop-Down Box Does Not Show Desired Option

- Use the scroll bar on the right to scroll down to the desired location.
- Verify that the desired option has been properly set up elsewhere.

Duplication of Cards or Alarms Does Not Work

- In Alarm Manager, alarms can be duplicated only for devices with the same profiles or similar points. See [Duplicating Alarms on page 120](#).

Errors on KMC Commander Web Pages

- See [“This Site Can’t Be Reached” Error on page 159](#).
- See [“Your Connection Was Interrupted” Error on page 161](#).
- See [“Your Connection Is Not Private” Error on page 161](#).

Ethernet Connection Indicators Are Unlit

- Check the wiring. See [Connecting Ethernet \[Advantech\]](#) or [Connecting Ethernet \[Dell\]](#).
- Verify the gateway is powered up. See [Connecting Power \[Advantech\]](#) or [Connecting Power \[Dell\]](#).
- Indicators can be hard to see in bright light. Temporarily shade them to check operation.

Graphics Stop Working or Screen Freezes

- A software update may be occurring. Try again later.
- Check communications.

KMDigital and KMD-5551E Translator Problems

- See [Configuring a KMDigital Network on page 38](#).
- Purchase a valid KMC Commander license for the KMD-5551E from KMC Controls (if not using with Niagara).
- Check that the KMD-5551E has been added to the network first.
- Check that the KMD-5551E device itself has been discovered and a device profile for it was assigned.

Note: If a device profile wasn't assigned, the translator did not receive the license refresh message sent from customer service. It therefore became unlicensed and stopped translating an hour after starting up. Complete for the KMD-5551E the processes described in [Discovering Devices on page 43](#) and [Assigning Device Profiles on page 43](#). You may also need to contact customer service to re-license the KMD-5551E translator.

- Check the KMD-5551E installation on the KMDigital and/or Niagara network.

- Check for network communication issues. See [Communication Problems \(Internet/Ethernet\)](#) on page 148.

Last Communication with Cloud (Yellow or Red) Triangle Appears

A triangle icon may appear on [app.kmccommander.com](#) if cloud communication with the gateway has been lost for:

- 5 minutes to 1 hour (yellow triangle).
- More than 1 hour (red triangle).
- See [Power Problems](#) on page 157.
- See [“500 Internal Error” on Login Screen](#) on page 147.
- See [Communication Problems \(Internet/Ethernet\)](#) on page 148.
- See [Communication Problems \(Cellular\)](#) on page 148.

License and Project Problems

Note: In the Cloud, project options are shown as **Project Name (license name for a gateway)**. Multiple KMC Commander IoT Gateways may be part of a single project, such as in; “My Big Project (IoT Box #1),” “My Big Project (IoT Box #2),” and “My Big Project (IoT Box #3).”

Note: A project license is entered by the project administrator in the Cloud account.

Note: **License Information** in **Settings > Project** shows the number of **Licensed Points** and **Used Points**. See [License Information](#) on page 12.

- If more than one license/project appears in the drop-down box during login, be sure to select the correct one for your particular project.
- If the correct license/project is not available in your login, contact your project administrator.
- Contact KMC Controls for a valid KMC Commander license.

Login Problems

Note: The System Administration server address moved to [central.kmccontrols.com](#) (from the older [projects.kmccontrols.com](#)) in November 2020.

- Verify that the correct browser address, user email and password are being used. See the KMC Commander Quick Setup guide (see [Support Documents and Resources](#)).

- If you are logging in with different credentials than a previous session, delete your browser history or use the browser's incognito mode.
- If the password is forgotten, click the Forgot password link, enter your email address, and you will receive an email with a link for resetting your password.
- See [Communication Problems \(Internet/Ethernet\) on page 148](#).
- Contact KMC Controls for assistance.

Modbus TCP Network Problems

Note: Unlike BACnet, only one Modbus TCP device is added to the “network” during discovery according to the device information entered. For multiple Modbus devices, create multiple Modbus “networks.”

- Ensure the proper register map for your device has been uploaded. See the [Modbus Devices on KMC Commander Application Guide](#) on the KMC web site
- See [Configuring a Modbus Network on page 40](#).
- Check for network communication issues. See [Communication Problems \(Internet/Ethernet\) on page 148](#).

Node-RED Problems

- See [Configuring a Node-RED Network on page 42](#).
- See [Node-RED Add-on on page 145](#).
- Check for network communication issues. See [Communication Problems \(Internet/Ethernet\) on page 148](#).

Password or User Name Not Recognized

- Select **Forgot password**, enter your email address, and you will receive an email with a link for resetting your password.
- If you never received login credentials, contact your system administrator or KMC Controls for assistance.

Points Do Not Drag-and-Drop

- In **Site Explorer**, drag the point to the virtual device and drop it when the green check mark (instead of a red NO symbol) shows. See [Creating a Site Topology on page 47](#).
- See [Points or Devices Not Visible in Site Explorer on page 155](#).

Points or Devices Not Visible in Site Explorer

- Under **Settings > Users/Role/Groups > Users**, check user permissions (Admin, etc.). See [Adding and Configuring Users on page 18](#).

- Check that the relevant boxes are all checked in user Topology Access in both the local box and cloud. See [Configuring a User's Topology Access on page 18](#).
- Refresh the browser.
- Select **Networks Explorer** again.
- Wait several minutes for Site Explorer to finish populating devices and points.

Points Needed for Cards or Trends Are Missing

- Select or create a correct device profile. See [Assigning Device Profiles on page 43](#).
- Edit the profile to include the points of interest.
- Select the correct **Type** for each of the selected points of interest.
- Verify that the points of interest are properly configured.
- If other software was used to add or change the name of a point on the device itself, rediscover the points and save the profile to show the changes.

Points Stopped Communicating

- Check that the device is not offline.
- Check the device configuration.
- See [Communication Problems \(BACnet MS/TP\) on page 147](#) and [Communication Problems \(Internet/Ethernet\) on page 148](#).
- See [Points Needed for Cards or Trends Are Missing on page 156](#) and [\(Card/Trend\) Point Values Are Missing or Incorrect on page 156](#).

(Card/Trend) Point Values Are Missing or Incorrect

Note: It may take a moment before all cards and present values are visible.

- Check that the desired point is really configured for the particular slot or trend (for example, that it isn't swapped with another point).
- Check that the connections, sensor(s), and controller(s) are functioning properly and are online.
- Check that there has not been a change in network configuration.
- Create a new profile from scratch, and use it instead of the old profile. See [Creating and Assigning a New Device Profile on page 44](#).
- Delete the affected card and/or trend and recreate it or them.
- See [Points Needed for Cards or Trends Are Missing on page 156](#).

“Point ____ has no data in this time window” Error

- If the error message appears for a new trend, allow time (a half hour or more) for initial point sampling and check the trend a later.
- If the point never appears, check the point’s configuration in the profile.
- If the point never appears in the trend but the profile seems correct, delete the profile and recreate it.

Power Problems

- For a 12–57 VDC power supply connected to the terminal block, see [Connecting with a DC Power Supply](#) [Advantech] or [Connecting Power with a DC Power Supply](#) [Dell].
- Verify that the Power indicator on the PoE injector is On (solid green). Ensure the Ethernet cables are connected to the correct ports on the PoE injector. See [Connecting with PoE \(Power over Ethernet\)](#) [Advantech] or [Connecting Power with PoE \(Power over Ethernet\)](#) [Dell].

Note: Do *not* connect an AC transformer to either connection on the gateway!

Note: When power is connected, the Power LEDs on both the PoE injector (if used) and the gateway should immediately glow green.

- If the Power LED on the gateway is amber instead of green, the power supply voltage is too low and the gateway will not be able to communicate. Verify that the PoE injector or other power supply is operating properly (at least 12 VDC and not more than 57 VDC). Try a different injector or power supply.
- If the Power LED on the PoE injector is amber and flashing slowly instead of glowing a solid green, try unplugging the power supply to it and plugging it back in. If that doesn’t help, replace the injector and its power supply.

Profiles Do Not Show

- KMC Conquest BAC-9xxx VAV and unitary controllers, BAC-8xxx SimplyVAV controllers, BAC-4xxxxxxxxx AppStats, and some other devices have a library of default profiles. Select the most applicable profile and modify as needed. See [Assigning Device Profiles on page 43](#).
- For general purpose controllers and other applications not included (no drop-down profile box appears for the device in **Edit Profile**), select **Create New** and create a custom profile.

Replaced Network Device Does Not Work

Note: If a bad device (for example, a BACnet controller) on a network is replaced, communication issues with the new device may result if any significant details are different from the old device.

- Ensure the new device’s model and name match what had been discovered with the old device.
- Ensure the new device’s model, point list, and point names match those in the assigned profile.

- Rediscover and reconfigure the new device and connected schedule, alarms, and trends.

Reset/Restore to Factory Defaults

Note: On rare occasions, the KMC Commander IoT gateway may need to be restored (reset) to its original factory default settings. This should only be done by the direction of KMC Controls Technical Support.

- See the *Restoring KMC Commander Factory Defaults Technical Bulletin* and contact KMC Controls for assistance. (See [Accessing Other Documents on page 164](#).)

Schedules (or Events) Do Not Work

- Check that the schedule is configured correctly and the schedule write priority is at an appropriate level (the default priority level value is 16). See [Managing Schedules and Events on page 94](#) and [Configuring Protocol Settings on page 13](#).
- Check that events are configured correctly and at the desired times.
- Delete the event or schedule and recreate it.
- To eliminate conflict with other schedules in a device, delete, set to NULL, or set to a lower priority level any schedules internal to the device or written by another device.
- To be able to command a point from the schedule or card, the point must be made commandable (by selecting it in the **Point Type** column and ensuring that **Point of Interest** is checked) in the device profile. See [Assigning Device Profiles on page 43](#).
- Inputs cannot be controlled directly. Map an input (e.g., from an STE-6020 dial setpoint on AI2 or a relay contact on B3) into an AV or BV and control the mapped value.
- To change the current setpoint, command the active heating/cooling setpoint rather than the (scheduled) occupied or unoccupied heating/cooling setpoint.

SIM or SD Card Cannot Be Inserted

Note: On KMC Commander Dell gateways only, the SIM card holder (upper slot) is for cellular connections. The SD card holder (lower slot) is not supported for KMC Commander.

- See [Installing Optional Cellular](#).
- Contact KMC Controls for assistance.

SNMP Network Problems

Note: Unlike BACnet, only one SNMP device is added to the “network” during discovery according to the information entered. For multiple SNMP devices, create multiple SNMP “networks.”

- See [Configuring an SNMP Network on page 40](#).

- See the documentation for the SNMP device.
- Ensure the correct MIB file (provided by the device manufacturer) has been loaded for the SNMP device.
- Edit the number for the Destination Port and Trap (notifications) Port if needed. (See the device's instructions.)
- Check user and security information. Security settings are typically found in the SNMP device's documentation or web management page. Use the highest security supported by the device.

Software Tools

Potentially useful network and installation troubleshooting tools include the following free software:

- [PuTTY](#)—an open-source terminal emulator, serial console, and network file transfer application. (Double-click the downloaded EXE file, enter the IP address of the gateway, and click Open. Select Yes if asked about trusting the certificate. Connect to the gateway using its SSH credentials. (See [Configuring SSH Settings on page 36](#).)
- [WinSCP \(Windows Secure Copy\)](#)—an open-source SFTP, FTP, WebDAV, and SCP client for Microsoft Windows that is used for secure file transfer between a local and a remote computer.
- [Wireshark](#)—an open-source packet analyzer used for network analysis and troubleshooting.
- [Simple IP Config](#)—an IP changing utility to quickly and easily change the most common network settings in Windows temporarily and then return to the original saved configuration.
- [Advanced IP Scanner](#)—a network scanner to analyze LANs (showing all network devices, providing access to shared folders, and providing remote control of computers).

Note: If you contact KMC for technical support, the representative might ask you to install and start up one or more of these tools to facilitate troubleshooting.

Temperature Scale (°F or °C) Is Wrong

- For the selection of the Fahrenheit or Celsius scale, see [Measurements on page 11](#) in **Settings > Project** and [Temperature on page 26](#) in **Settings > Weather**.

“This Site Can’t Be Reached” Error

- If the page took too long to respond, wait a few minutes and refresh the page in your browser.
- See [Communication Problems \(Wi-Fi\) on page 150](#) and [Communication Problems \(Internet/Ethernet\) on page 148](#).

Time is Wrong

- Check the time and date settings. See [Configuring Date and Time Settings on page 31](#).

Trends Do Not Work

- Refresh the graph. See [Viewing Trends on page 107](#).
- Check that the trend is configured correctly. See [Creating Trend Graphs on page 102](#).
- On a card, select no more than four points per trend.
- See [Communication Problems \(BACnet MS/TP\) on page 147](#), [Communication Problems \(Internet/Ethernet\) on page 148](#), [Communication Problems \(Cellular\) on page 148](#), and/or [Communication Problems \(Wi-Fi\) on page 150](#).

Units of Measurement or Temperature Are Wrong

- For the selection of units of measurement across the whole project, see [Measurements on page 11](#) in **Settings > Project**
- For the selection of temperature units for Weather cards, see [Temperature on page 26](#) in **Settings > Weather**.

Virtual Point is “Undefined”

- For some points (e.g., occupancy mode), the sum and average options might be “undefined” and invalid. See [Creating a Virtual Point on page 48](#).
- A valid virtual point may show as “undefined” until it has accumulated data for calculations. Refresh the screen to see the results.

VPN Problems

Note: This assumes you can access the interface locally through Ethernet, but not through a VPN connection. See also [Communication Problems \(Internet/Ethernet\) on page 148](#).

- Add the VPN address to the whitelist in **Settings > Whitelist/Blacklist**. See [Configuring Whitelist/Blacklist Settings on page 33](#).
- Contact the building’s IT department for assistance with the correct VPN login information, opening of any relevant internal firewalls, and other networking issues.

Weather Card Problems

Note: In software version 2.0, the weather station location was selected on the Dashboard card. In version 2.1 and after, one or more weather stations are selected in **Settings > Weather** first, and then which of those stations is shown on that card is selected from a drop-down box during the card’s configuration. See [Configuring Weather Settings on page 26](#) and [Creating a Weather Card on page 68](#).

- If a desired weather station location is not visible in the drop-down list, use the scroll bar on the right to scroll down to the desired location.

- Verify that the desired locations have been saved in **Settings > Weather**.
- If the selection of °F or °C is wrong, change it in **Settings > Weather**.

Web Card Problems

- A desired web page must be HTTPS with a public URL (no on-premise IPs), the link must be able to be embedded, and the site must allow HTML Inline Frame (iframe) elements. See [Creating a Web Card on page 69](#).

“Your Connection Was Interrupted” Error

- If this message with “A network change was detected” occurs in the browser screen, the Wi-Fi network may have a higher priority for Internet access than the LAN connection. Change the network priority. (See Google for possible solutions.)

“Your Connection Is Not Private” Error

- This message may occur in the browser screen when first signing into the gateway through Wi-Fi and/ or Ethernet. Select **Advanced**, then **Proceed**. See [Connecting Wi-Fi and Making Initial Login](#).

Restarting the Gateway

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Gateway Restart Options

The KMC Commander IoT gateway can be restarted either remotely or locally by doing one of the following procedures:

- In **Settings**  > **Project** > **Gateway Information**, select **Reboot Gateway**. (See [Gateway Information on page 12.](#))

Note: The gateway must have a Cloud connection to perform a remote reboot.

- Press the power reset button (Advantech only) off and on.
- Connect to the gateway with PuTTY or equivalent terminal software, using its SSH credentials, and at the ~\$ prompt, enter sudo reboot. (See [Configuring SSH Settings on page 36.](#))
- Temporarily disconnect power to the gateway. For details, see [Connecting Power](#) (Advantech) or [Connecting Power](#) (Dell).

After restoring power, allow at least three minutes for the gateway to fully reboot. Then log back into the gateway.

Software Updates

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

About Software Updates

KMC Commander software automatically updates the project cloud (app.kmccommander.com) and the IoT gateway hardware. Users do not need to do anything.

For lists of features in the current and previous software releases, see the [Version History](#) online at help.kmccommander.com.

Accessing Other Documents

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

How to Access Other Documents

To find information located in other documents:

1. Go to the [KMC Controls](#) website and **Sign In** (upper-right corner) to the **Partner Portal**.

Note: Signing in allows you to see all available documents. You can access the [Data Sheets on page 165](#), [Installation Guides on page 165](#), and most of the items under [Application Guides on page 164](#) without signing in.

2. Go to any KMC Commander product page (found under [Base IoT \(Internet of Things\) Packages](#)).
3. Select the **DOCUMENTS** tab (at the bottom of that page).
4. Locate the document you need by expanding its category.

Note: To find the category to which the document belongs, consult the lists below.

Note: To find documents specific to the discontinued KMC Commander Dell gateway: (After logging into the partner portal) go to the [Support > Downloads](#) page, then search for the title of the document (consult the lists below).

Application Guides

- *Commander Sample Modbus Register Map Files*
- *KMCAFMS with KMC Commander Application Guide*
- *KMC Commander (Cloud) System Administration* (must sign in to view)
- *KMC Commander and Node-RED*
- *KMC Commander for Growers*
- *KMC Commander Software Application Guide*
- *Modbus Devices on KMC Commander*

Bulletins

- *KMC Commander IT Network Information*

Note: Information about how to access the offline login page is given in the *KMC Commander IT Network Information* bulletin under "Testing Cloud Connections".

- *KMC Commander Thermostat Card Change*
- *Restoring KMC Commander Factory Defaults (Advantech)*
- *Restoring KMC Commander Factory Defaults (Dell)*

Data Sheets

- *KMC Commander (Advantech) IoT and Automation Platform*
- *KMC Commander (Dell) IoT and Automation Platform*
- *KMC Commander Gateway Service for Niagara 4*
- *KMC Commander Virtual Machine*

Guide Specifications

- *2022 KMC Master Guide Specification - Part 1 - General - Commander IoT Solution*
- *2022 KMC Master Guide Specification - Part 2 - Section 2-1 KMC BACnet System Architecture (KMC Commander Cloud)*
- *2022 KMC Master Guide Specification - Part 2 - Section 2-5 - Commander*
- *KMC Commander Guide Specification E*

Installation Guides

- *KMC Commander (Advantech UNO-420) Installation Guide*
- *KMC Commander (Advantech UNO-420) Quick Setup Guide*
- *KMC Commander (Dell) Installation Guide*
- *KMC Commander (Dell) Quick Setup Guide*
- *KMC Commander Gateway Service for Niagara Installation Guide*

Contact Information

KMC Controls, Inc.
19476 Industrial Drive
New Paris, IN 46553
U.S.A.
TEL: 1.574.831.5250 or 877.444.5622
Email: info@kmccontrols.com
Web: www.kmccontrols.com

Important Notices

Do you find this article helpful? Help us improve by sharing feedback: [Give Feedback](#)

Trademarks

KMC Conquest™, KMC Controls®, and the KMC logo are registered trademarks of KMC Controls, Inc. All other products or name brands mentioned are trademarks of their respective companies or organizations. All rights reserved.

Patents

Pat. <https://www.kmccontrols.com/patents/>

Terms of Use

<https://www.kmccontrols.com/terms/>

Copyright

No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form by any means without the written permission of KMC Controls, Inc.

Disclaimers

The material in this document is for information purposes only. The contents and the product it describes are subject to change without notice. KMC Controls, Inc. makes no representations or warranties with respect to this document. In no event shall KMC Controls, Inc. be liable for any damages, direct or incidental, arising out of or related to the use of this document.