WinPM.Net 10.0

User Guide

07/2021

SIEMENS

Safety Information

Important Information

Read these instructions carefully and look at the equipment to become familiar with the device before trying to install, operate, service or maintain it. The following special messages may appear throughout this bulletin or on the equipment to warn of potential hazards or to call attention to information that clarifies or simplifies a procedure.

The addition of either symbol to a "Danger" or "Warning" safety label indicates that an electrical hazard exists which will result in personal injury if the instructions are not followed.

This is the safety alert symbol. It is used to alert you to potential personal injury hazards. Obey all safety messages that follow this symbol to avoid possible injury or death.

⚠ DANGER

DANGER indicates a hazardous situation which, if not avoided, **will result in** death or serious injury.

⚠ WARNING

WARNING indicates a hazardous situation which, if not avoided, **could result in** death or serious injury.

⚠ CAUTION

CAUTION indicates a hazardous situation which, if not avoided, **could result in** minor or moderate injury.

NOTICE

NOTICE is used to address practices not related to physical injury.

Please Note

Electrical equipment should be installed, operated, serviced and maintained only by qualified personnel. No responsibility is assumed by Siemens Industry for any consequences arising out of the use of this material.

A qualified person is one who has skills and knowledge related to the construction, installation, and operation of electrical equipment and has received safety training to recognize and avoid the hazards involved.

Notices

	⚠ DANGER
	Hazardous voltages and high-speed moving parts in electrical devices communicating with WinPM.Net can cause death, serious injury or property damage. See Safety instruction contained herein. **Restrict use to qualified personnel**.

The use of unauthorized parts in the repair of the equipment or tampering by unqualified personnel will result in dangerous conditions that can cause death, serious injury or property damage.

PLEASE READ

WinPM.Net can be configured to send commands to devices automatically. WinPM.Net can be used to manually send commands to devices and to reconfigure the settings of devices.

IMPORTANT

The information contained herein is general in nature and not intended for specific application purposes. It does not relieve the user of responsibility to use sound practices in application, installation, operation, and maintenance of the equipment purchased. Siemens reserves the right to make changes at any time without notice or obligations. Should conflict arise between the general information contained in this publication and the contents of drawings or supplementary material or both, the latter shall take precedence.

QUALIFIED PERSONNEL

For the purposes of this manual and product labels, "qualified personnel" is one who is familiar with the installation, construction, or operation of the equipment and the hazards involved. In addition, s/he has the following qualifications:

(a) **is trained and authorized** to energize, de-energize, clear, ground, and tag circuits and equipment in accordance with established safety practices.

(b) **is trained** in the proper care and use of protective gear equipment such as rubber gloves, hard hat, safety glasses or face shields, flash clothing, etc., in accordance with established safety procedures.

(c) **is trained** in rendering first aid.

SUMMARY

These instructions do not purport to cover all details or variations in equipment, nor to provide for every possible contingency to be met in connection with installation, operation, or maintenance. Should further information be desired or should particular problems arise which are not covered sufficiently for the purchaser's purposes, the matter should be referred to the local Siemens Industry, Inc. sales office. THE CONTENTS OF THIS INSTRUCTION MANUAL SHALL NOT BECOME PART OF OR MODIFY ANY PRIOR OR EXISTING AGREEMENT, COMMITMENT OR RELATIONSHIP. THE SALES CONTRACT CONTAINS ALL OBLIGATIONS OF SIEMENS INDUSTRY, INC. THE WARRANTY CONTAINED IN THE CONTRACT BETWEEN THE PARTIES IS THE SOLE WARRANTY OF SIEMENS INDUSTRY, INC.

ACCESS, ISGS, Isolated Multi-Drop, S7-I/O, SBwin, SAMMS-LV, SAMMS-MV, SEAbus, SIEServe, Static Trip III, Wisdom, and WinPM.Net are trademarks, Sensitrip and Sentron are registered trademarks of Siemens Industry, Inc. SIEMENS is a registered trademark of Siemens AG. Windows is a trademark of Microsoft Corporation. ION, Modbus and WebMeter are either trademarks or registered trademarks of Schneider Electric in France, the USA and other countries. All other product names mentioned herein are used for identification purposes only and may be the trademarks or registered trademarks of their respective companies.

Contents

Safety Precautions

During installation or use of this software, pay attention to all safety messages that occur in the software and that are included in the documentation. The following safety messages apply to this software in its entirety.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software or devices for critical control or protection applications where human or equipment safety relies on the operation of the control action.
- Do not use the software to control time-critical functions.
- Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

⚠ WARNING

POTENTIAL COMPROMISE OF SYSTEM AVAILABILITY, INTEGRITY, AND CONFIDENTIALITY

Use cybersecurity best practices to help prevent unauthorized access to the software.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

Work with facility IT System Administrators to ensure that the system adheres to the site-specific cybersecurity policies.

Chapter 1: Introduction to WinPM.Net

WinPM.Net software is a power management solution for energy suppliers and consumers. It allows you to manage energy information from metering and control devices installed in your facility or other remote locations. The product offers control capabilities and comprehensive power quality and reliability analysis to help you reduce energy-related costs.

The product supports multiple communications standards and protocols available on various intelligent metering devices. You can also connect to existing power monitoring systems through industry standard protocols such as Modbus and OPC.

Supported operating systems and SQL Server versions

See Operating Environment for information on supported Windows operating systems and SQL Server versions and editions.

WinPM.Net software components

This section outlines the WinPM.Net software components.

Web Applications

Web applications are the end user facing components of the software. You use web applications in your daily work with WinPM.Net. Web Applications has 3 main parts: Apps, Settings, and Configuration Tools.

Apps

Use Web apps to access power monitoring information. The following apps are included in the Web Applications:

- Dashboards
- Diagrams
- Trends
- Alarms
- Reports

(See Web Applications help for details.)

Settings

Use Settings to tailor the behavior and appearance of the software. The following Web Application Settings are available:

- Alarm Views
- Authorized Hosts
- Diagrams Control Options
- Login Options
- Personal Preferences
- Report Theme

- Security Options

- Session Timeout

- System Language

- System Theme

The Settings page also provides links to the following configuration tools:

- Software Alarms

- Device Manager

- Hierarchy Manager

- Log Viewer

- Notifications

- User Manager

(See the individual tool helps for details.)

Engineering applications

Engineering applications are the backend components that are used to configure and maintain the monitoring devices, networks, databases and other elements of the power monitoring system. It also includes the Vista application which is used to for real-time control and to build diagrams. The following applications are included:

Management Console

Use Management Console to add and configure network components such as servers, sites (communication links) and devices. Management Console also provides access to the following system and database applications, and utilities (listed in menu sequence):

Through **Management Console > Tools**

- [Database Manager](#)

- [Event Watcher Manager](#)

- [Reports Configuration](#)

- [Logical Device editors](#)

- [Deactivating alarms](#)

Through **Management Console > Tools > System**

- [Diagnostics Viewer](#)

- [Device Type Editor](#)

- [Manual Data Editor](#)

- [Remote Modem Setup](#)

- [Virtual Processor setup](#)

- [Update OPC Server](#)

- [Upload Report Template](#)

- [PQDIF Exporter](#)
- [Time of Use Editor](#)

Through **Management Console > Tools > Web Tools**

Management Console provides shortcuts to Web Applications configuration tools. See [Web Applications](#) for details.

Vista

Use Vista to reset counters and perform other control actions on monitoring devices. Vista also displays real-time and historical information using graphical displays. You use Vista to create the diagrams for the Diagrams Web app.

Designer

Use Designer to perform a wide range of functions, from configuring setup registers of ACCESS devices on your network to creating complex frameworks using a combination of ION modules from hardware or software nodes.

OPC Server Assistant

The OPC Server Assistant is a utility that you use to expose ION measurements as OPC tags. You open the utility from Designer. See [OPC Server Support](#) for further information.

WinPM.Net can be ordered with or without the optional OPC server license. You can also order the OPC server license at a later date. If you purchased your OPC server license after you have installed WinPM.Net, enable the OPC server functionality simply by entering your new product key in Management Console (under **Tools > License Manager**).

Large system setup using Management Console

Use the Duplicate and Configure function to set up large systems efficiently. See [Setting up large systems](#) for more information.

WinPM.Net Windows Services

For a list of WinPM.Net Windows Services see [WinPM.Net Windows services](#)

WinPM.Net databases

For information on the WinPM.Net databases, see [WinPM.Net DatabasesWinPM.Net Databases](#).

Getting started

This section describes how to start, log on to, and exit most of the components of the WinPM.Net product.

Starting a component of WinPM.Net

Use one of the following methods to start the main components Management Console, Vista, or Web Applications of your product installation:

- Open the WinPM.Net folder on your desktop and double-click the icon of the program you want to start.

- Click **Start > All Programs > Siemens > WinPM.Net** and select the program you want to start.

Logging into a component

Most WinPM.Net programs require you to log in before you can access them.

At the logon prompt, type your user name and password in the appropriate box, then click **OK**. Some actions are restricted to certain users or groups, depending on how their login permissions were set up. For information on creating users and user groups, and on setting user access levels, see the User Manager section in Web Applications Help.

NOTE: For security reasons, using **View Only** button to login to Vista no longer works and displays an error message. Alternatively, it is recommended to login to Diagrams application with Observer access level.

After you log on, the program interface appears.

Ending a session

There are two ways to end a session in a component:

- **Log off** if you want to end the current user session but keep the program running, or
- **Exit** if you want to close the program completely.

Some components offer both options; some only offer the Exit option.

Logging off

1. Select **File > Logoff**.

   A message appears, prompting you to confirm your intention to log off.

2. Click **Yes** to log off or **No** to return to the program.

   If you attempt to log off without saving your work, you are prompted to save your changes. Click **Yes** to save your changes, **No** to discard them, or **Cancel** to return to the program.

   After you log off, the component's **Logon** screen appears.

Exiting

1. Select **File > Exit**.

   A message appears, prompting you to confirm your intention to exit the application.

2. Click **Yes** to exit or **No** to return to the program.

   If you attempt to exit without saving your work, the program prompts you to save your changes. Click **Yes** to save your changes, **No** to discard them, or **Cancel** to return to the program.

Customizing and navigating interface displays

The following sections describe ways to customize and navigate display windows, dialogs, and tables in components of WinPM.Net.

Navigating tree structures

Information is often displayed in a tree structure. Click "+" to expand items in the tree or "-" to collapse them.

Selecting multiple items in tables, tree structures, diagrams and folder structures

To select adjacent items, select the first item, hold down the SHIFT key then click the last item.

To select non-adjacent items, hold down the CTRL key then click to select the items.

To select adjacent items in Vista or Designer diagrams, drag a selection box around the items. All items in the box are selected.

Adjusting the display of a window or dialog

The following sections describe how to resize and hide or reveal panes.

Resizing panes

To resize panes in a window, point the mouse at the border where the two sections meet. When the resize handle appears, drag to move the border.

Dock (Pin) or hide (Unpin) pane

Some component interfaces have panes that can be hidden (visible temporarily) or docked (visible all the time).

To hide or dock a pane, click the Pin/Unpin ⬚ button on the pane. When a pane is hidden, it is minimized to a button at the edge of the workspace.

To temporarily show a hidden pane, hover the mouse over the button for that pane. Move the mouse away to hide the pane again.

Grid table controls

Certain windows and utilities display information in "grid" tables. These tables have unique table and information display commands.

Select columns/column selector

Some tables allow you to select which columns to include in the table.

1. In Management Console, hover over the column selector tab on the right side of the display window. In other interfaces, click the "Select Columns" link.

2. Select the boxes for the columns you want to include in the table. Clear the boxes for the columns you do not want to include in the table.

Adjust column width

To change the width of a column, point the mouse to the right edge of the column header. When the resize handle (double-arrow pointer) appears, click and drag to adjust the width. To adjust the width for best fit, point to the right edge of the column header, then double-click when the resize handle appears.

Best Fit

To adjust a column width to the best fit with the least amount of white space:

- For one column, right-click a column header and select **Best Fit**.

- For all columns, right-click the column title area and select **Best Fit (all columns)**.

Arrange columns

To change the order of the columns, drag a column header to the left or right of its original position.

Group by column

To group data according to the contents of a particular column, drag the column header to the area above it (marked "Drag a column header here to group by that column"). To expand or collapse the groups, click the "+" or "-" button. To expand or collapse all groups, right-click the column header in the Group By Box area and select **Full Expand** or **Full Collapse**, respectively.

You can also group by multiple columns in a specific order, with sorting and filtering applied. To do this drag each column to the **Group By Box** area in the order that you want to group them.

To ungroup, drag the column header back to its original position (or right-click the column header and select **Ungroup**).

To show or hide the **Group By Box** area, right-click the column header and select or clear **Group By Box**.

Sorting data in a column

Click a column header to sort table rows according to data in that column. You can sort in ascending or descending order (indicated by an up or down arrow, respectively). To cancel sorting and return the column to its default state, right-click the column and select **Clear Sorting**.

Filtering data in a column

The following sections describe the available filtering options.

Using the Simple Column Filter

When the mouse is positioned over a column header, the Filter icon appears in the top right corner of the header. Click it to select one of the listed filter conditions and apply it to the data in that column. Select **(Custom)** to apply one or two logic conditions to filter the data. To cancel filtering and return the column to its default state, right-click the column and select **Clear Filter**.

Using the Dynamic Column Filter

Some tables have a row between the column header and the table data. This is the dynamic filter area.

To use the dynamic filter area, start typing the entry you want to filter on in the space above the applicable column. You can use an asterisk (*) at the beginning of the filter as a wildcard. Once you achieve the filter results you want, you can stop typing. For example, to filter out "DST" from "disturbance" in System Log Events, you only need to type "di".

Using the Filter Builder

The Filter Builder is an advanced data filtering tool. Use it to create filters with multiple conditions across multiple columns. To use the Filter Builder:

1. Right-click a column header and select **Filter Editor** to open the **Filter Builder** dialog.

2. Specify the filter conditions.

    The elements of the filter builder are: <Column Header> <Condition> <Value>

    Click an element to display the options available, then select the option you want to use.
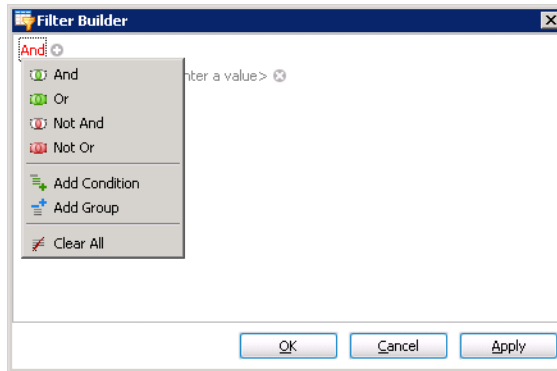
3. Select the <Column Header> and the <Condition> you want to apply. Type the <Value> to test for.

4. To add another filter, click the "+" button on the top.

5. Select a logic to apply to this new filter (in relationship to the current filter).

6. Repeat step 3.

7. To remove a filter, click the delete ("**x**") button beside it.

In the following example, the filter finds all devices in group name containing "Mfg" **and** whose types begin with "ION".

The rows returned are as follows:

A	Select the checkbox to turn the filter on; clear it to turn the filter off. Click the "X" to cancel and exit the filter mode.
B	Click **Edit Filter** to configure the filter conditions

Chapter 2: Management Console

Use Management Console to add, remove or configure system components, such as metering devices, in your WinPM.Net power management system. You can set up different types of sites (communication links such as Ethernet or serial), and set up connection schedules for these sites.

In addition, the Management Console **Tools** menu provides access to device and system configuration, maintenance and programming tools.

> **NOTE**: Run Management Console as Administrator user. To do this, right-click the Management Console program icon and select **Run as administrator** from the context menu. If you do not run it as Administrator, then certain tools are not functional.

NOTICE

NETWORK INOPERABILITY

Do not make unauthorized changes in the network configuration.

Failure to follow these instructions can result in an unstable or unusable network.

The Management Console interface
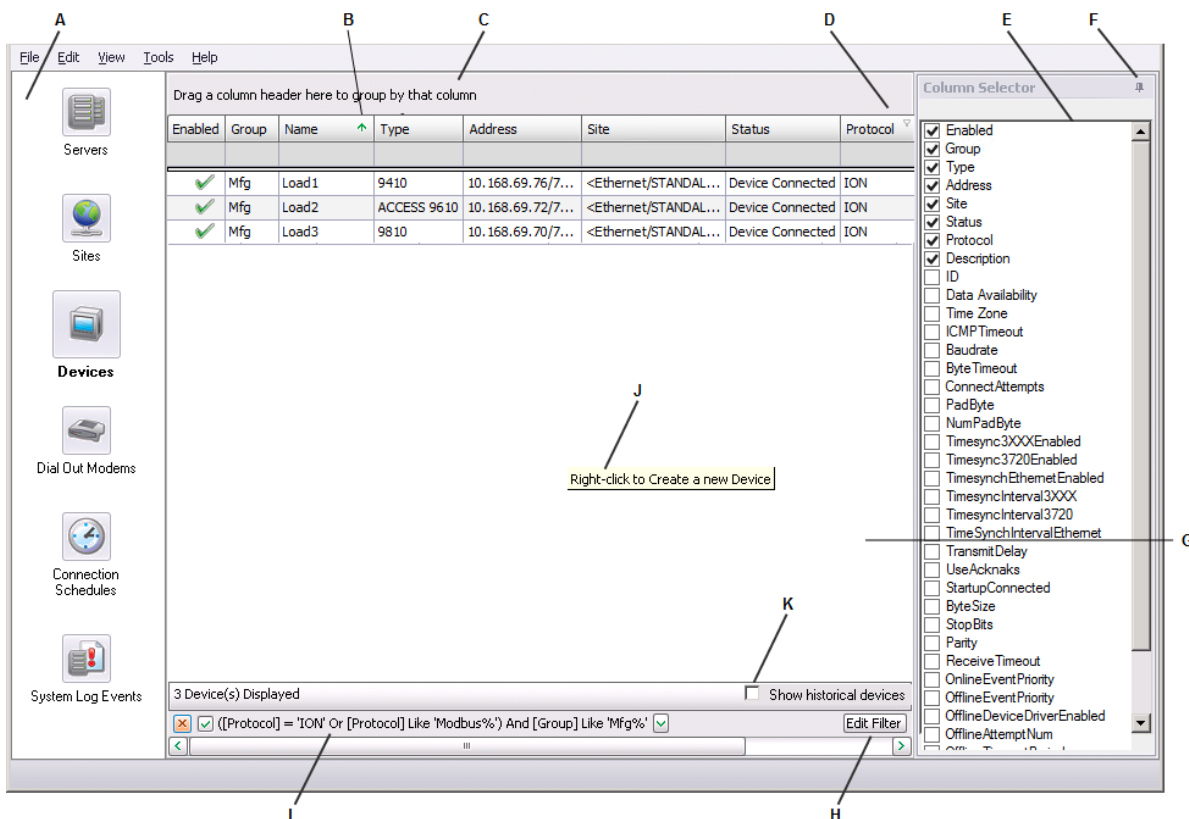
A	System Setup pane and icons	B	Sort indicator	C	Group By Box area	D	Filter indicator
E	Column Selector	F	Pin/Unpin icon	G	Display window[1]	H	Edit Filter button
I	Current filter applied	J	Hover text	K	Show historical devices option		

[1] This area displays the contents of the selected System Setup icon

Window title

The window title displays the name of the software program and the current user.

System Setup pane

The icons in the System Setup pane represent categories of items you can add, remove, set up, and view. Click an icon to go to the associated display window.

Display window

The Display window shows the contents of the item currently selected in the System Setup pane.

Refresh

To manually refresh Management Console and reload its contents, select **View > Refresh** (or press F5).

Options

Select **Tools > Options** to open the Options dialog. Use this dialog to change display settings for the System Log Events display window.

Highlighting

Highlighting controls how Error events are displayed in the System Log Events display window. Select or clear the Enable Highlighting check box to turn highlighting on or off. Click **Set Highlight Color** to change the highlight color for Critical or Error events in the System Log Events display window.

Events

The Events section of the dialog lets you control how many records appear in the System Log Events display window and how date and time are displayed.

Customizing the Management Console interface

You can customize what is displayed in the Management Console interface and the tables within the display windows:

- Select which columns appear in tables.
- Filter, sort and group data in tables.

For instructions on using these features, see [Customizing and navigating interface displays](#).

Display window shortcut menus

To access shortcut menus in the Display window, first click the icon of the System Setup item you are interested in. Then right-click the display window area or on a network component to display the shortcut menu. The menu items available vary depending on the network component

selected.

- **New** displays the options available for adding new network items.

- **Duplicate and Configure** lets you perform batch cloning of devices. When multiple devices are selected, this right-click shortcut becomes **Configure Selected Devices** which lets you perform setup functions such as batch renaming of devices.

- **Delete** removes the selected items.

- **Connect** and **Disconnect** lets you manually connect to or disconnect from the selected site.

  > **NOTE**: You cannot connect or disconnect an individual serial device in a site; you can only connect or disconnect the site for that serial device.

- **Connect Ethernet** and **Disconnect Ethernet** let you manually connect to or disconnect from the selected Ethernet device.

- **Security** accesses the Meter Security Settings dialog. This is used by the software to gain access to security-enabled devices (for example, ION8600 or 9610); it is only available from the shortcut menu for the Devices display pane.

- **Configure <Network Component>** opens the <Network Component> Configuration dialog (where <network component> is the component selected). Use this dialog to make changes to the properties for the selected network component.

For Devices only:

- **Rename** opens the Rename Device dialog. Use this to change the device Name.

- **Change Group** opens the Change Group dialog. Use this to change the device Group.

Management Console network components

This section describes the different components in Management Console: Servers, Sites, Devices, Dial Out Modems, Connection Schedules, and System Log Events.

See [Setting up your network in Management Console](#) for information on adding these components to your network.

> **NOTE**: Do not use names such as CON, AUX, COM1, and LPT1 when naming sites and devices. Consult Microsoft documentation regarding restrictions for naming files, paths, and namespaces.

Servers

A server is a computer in the WinPM.Net system that runs administrative software to control access to the network and its resources, such as metering devices.

Sites

A site is a group of devices in the system that share a common communications link. A site can be a direct site, a modem site, an Ethernet gateway site, or an OPC site.

Direct site

In a direct site, serial communications occur between a computer and one or more meters. The standards most commonly used on computers are RS-232 (for connecting one device) and RS-485 (for connecting a loop of up to 32 devices). When connecting to more than one serial device, use an RS-232 to RS-485 converter. Note that you should keep the number of devices on a daisy chain to a minimum to reduce the delay in real-time updates.

Modem site

In a modem site, communications occur between a remote modem (at the Modem Site) and a local modem (on the server).

NOTE: Remote sites that use radio modems or leased-line modems are configured as Direct (Serial) Sites, not as Modem Sites.

Modem sites can include traditional hardware modems (those that are configured and controlled by WinPM.Net or WinModems (Windows modems). If multiple modems are set up in the modem site, WinPM.Net selects the first available modem to establish communications (this is referred to as "modem pooling").

Ethernet gateway site

An Ethernet gateway site (for example, EtherGate or Modbus gateway) consists of an RS-485 chain of devices that communicates with an Ethernet network via an Ethernet device. The Ethernet device acts as a gateway that transfers data between an Ethernet network and the devices connected to it via RS-485. The gateway device converts RS-232/RS-485 communications to and from Ethernet.

OPC site

An OPC site consists of an OPC client that communicates with OPC-compliant devices on the network.

Devices

A device is a meter or other component that communicates and gathers data, and is installed on a network.

Release notes and mapping spreadsheets for device drivers recently added to the product are included in the product's installation location in `...\WinPM.Net\system\doc\Device Support\`. The release notes provide information about the firmware versions supported by the device drivers, implementation details, images of Vista diagrams for the devices, and pertinent safety messages. The device driver mapping spreadsheets contain details such as modbus register addresses and corresponding ION register addresses (handles). Register mapping information is also provided for any devices using PC logging.

Serial devices

Serial devices belong to direct sites, modem sites, or Ethernet gateway sites — they communicate through a modem, RS-232, RS-485, or Ethernet-to-Serial gateway connection. You must have the appropriate sites configured first before you can add serial devices.

Ethernet devices

Ethernet devices are those that are directly connected to the network via Ethernet. You can manually connect or disconnect your Ethernet device through the Management Console.

Logical devices

Logical devices are a collection of measurements from physical and VIP devices grouped into single sources for use in the system.

OPC devices

OPC devices represent physical devices to which the system is communicating through an OPC interface.

Dial Out Modems

A dial-out modem is one that a server computer uses to communicate with a remote modem at a modem site. The modem implemented can be a traditional hardware-based modem or it can be a software-based modem that uses the computer's processor to emulate a traditional hardware-based modem.

When a modem site needs to establish communications, it selects a modem from the list of modems configured in the modem site. You can add hardware modems or WinModems to the modem list.

NOTE: A WinModem is a modem that is directly controlled by the computer's operating system.

Connection schedules

Connection schedules are programmed routines for a server to regularly connect to and disconnect from sites (and their associated devices) and Ethernet devices. See Configuring connection schedules for more information.

System Log Events

System Log Events lists events for the network. You can select what time frame and components you want to view events for. See System Log events for more information.

Setting up your network in Management Console

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

⚠ **WARNING**
UNINTENDED EQUIPMENT OPERATION   • Do not use the software to control time-critical functions.   • Do not use the software to control remote equipment without proper access control and status feedback.   **Failure to follow these instructions can result in death, serious injury, or equipment damage.**

NOTICE
NETWORK INOPERABILITY   Do not make unauthorized changes in the network configuration.   **Failure to follow these instructions can result in an unstable or unusable network.**

The Primary Server contains all the WinPM.Net programs and it controls the overall operation of your energy management system. The primary server is displayed on the Server screen when you first start Management Console.

To start building your system, add and configure your sites and devices.

Overview

Site and Device Naming restrictions

The following applies to Site, Device Group, and Device Name naming:

- Names cannot contain spaces or the following characters: \ / : * ? " < > { } . , ' & @ | % #

- The maximum length for a `Group.Device` name is 99 characters (+ 1 for the period separation).

- Do not use names such as CON, AUX, COM1, and LPT1 for sites and devices.

- All characters must exist in the system's Windows code pages. For example. on an English Windows operating system, certain non-English language characters are not supported.

Adding network components

For details on adding various components, see the following sections.

Follow the procedure below to add a new network component:

1. In the System Setup pane, select the icon for the type of item that you want to add (**Site**, **Device**, or **Dial Out Modem**).

2. Right-click the display window, select **New**, and then select the specific type of network component from the shortcut menu to open the **Configuration** dialog.

   The options differ depending on which system setup icon you selected.

3. Use the fields and drop-down menus to configure properties. Mandatory items are highlighted in red. To configure advanced properties, right-click the dialog and select **Advanced Properties**. For more shortcut menu options, see Configuration Dialog shortcut menus, below.

4. Click **OK** to add the component.

Configuration Dialog shortcut menus

Right-click the **Configuration** dialog to access the following shortcut menu options:

- **Property Details**: This opens a window that contains information on the selected property, including minimum and maximum allowable values, if applicable.

- **Advanced Properties**: This exposes all the property fields that are hidden in the default (basic) view. For further information about the Time synchronization option. see [Time synchronization for SIEMENS 9xxx series meters](#) for further information.

- **Reset All To Default**: Resets all properties to their original values.

Adding a server

Click the Servers icon, right-click the display window and select **New > Computer**. Fill in the mandatory **Name** field.

Adding devices

Setting Up devices for direct serial communications

To set up devices for RS-232 or RS-485 communications on a direct serial loop, first add a direct site in Management Console and specify a serial communications port on the computer.

Step1: Adding a direct site

Click the Sites icon. Right-click the display window and select **New > Direct Site**. Fill in the **Name** and **Serial Port** fields. Configure the other fields as you require.

Step 2: Adding a device to the direct site

Click the Devices icon. Right-click the display window and select **New > Serial Device** on **Direct Site**. Fill in these fields:

- **Group**: Type a name for the group or select an existing group from the list.

- **Name**: Type the name you want to give the device.

- **Device Type**: Select the type of the device.

- **Unit ID**: Type the Unit ID of the device; the range is 1-9999 for SIEMENS 9xxx series devices or 1-247 for Modbus RTU devices.

- **Site**: Select the direct site you previously set up.

- **Time Zone**: Select the timezone you want the device data to be displayed in, in the software. This setting is only used for the display of timestamped data in the software. It does not affect the configuration of the monitoring device itself.

Configure the other fields as you require.

Setting up devices for Ethernet gateway communications

To set up devices for Ethernet gateway communications, first add and configure an Ethernet gateway site. Ethernet gateways that can be used include those that are simple Ethernet-to-Serial converters, or any gateway or data concentrator that provides ModbusTCP communications.

Step 1: Adding an Ethernet gateway site

Click the Sites icon. Right-click the display window and select **New > Ethernet Gateway Site**.

Complete the **Name**, **IP Address or Host Name** and **TCP/IP Port** fields. (The IP address can be IPv4 or IPv6.) The TCP/IP Port identifies which communications port is used to connect to the serial devices. The port you use depends on the type of Ethernet gateway you want to set up (use 7801 for an EtherGate gateway via COM1 or use port 502 for a Modbus gateway).

Configure the other fields as you require.

Step 2: Adding a Device to the Ethernet Gateway Site

Click the Devices icon. Right-click the display window and select **New > Serial Device on Ethernet Gateway Site**. Fill in these fields:

- **Group**: Type a name for the group or select an existing group from the list.
- **Name**: Type the name you want to give the device.
- **Device Type**: Select the type of device.
- **Unit ID**: Type the Unit ID of the device; the range is 1-9999 for SIEMENS 9xxx series devices or 1-247 for Modbus devices.
- **Site**: Select the Ethernet gateway site you previously set up.
- **Time Zone**: Select the timezone you want the device data to be displayed in, in the software. This setting is only used for the display of timestamped data in the software. It does not affect the configuration of the monitoring device itself.

Configure the other fields as you require.

Adding an Ethernet device

Ethernet devices are configured in WinPM.Net by providing fixed IP addresses (IPv4 or IPv6) and ports, or based on device names. Device names must be used for devices with dynamic address assignment, for example using the DHCP protocol. When device names are used, then a form of name resolution mechanism is required on the network.

To add an Ethernet device, click the Devices icon. Right-click the display window and select **New > Ethernet Device**. Fill in the **Group**, **Name**, **Device Type**, **IP Address or Host Name**, and **Time Zone** fields. (The IP address can be IPv4 or IPv6.) Configure the other fields as you require.

NOTE: For the **Time Zone** field, select the timezone you want the device data to be displayed in, in the software. This setting is only used for the display of timestamped data in the software. It does not affect the configuration of the monitoring device itself.

Setting up devices for OPC communications

To set up devices for OPC communications, first add an OPC site.

Step 1: Adding an OPC site

Click the Sites icon. Right-click the display window and select **New > OPC Site**. Fill in the **Name** and **Address** fields. Configure the other fields as you require.

Make sure you follow the correct syntax when typing in the address for the OPC site. Select the **Address** field to see an example address in the Property Description section (near the bottom of the **OPC Site Configuration** dialog).

Step 2: Adding a device to the OPC site

NOTE: At least one OPC Device Type needs to be preconfigured before an OPC Device can be created in Management Console. Contact Technical Support for assistance.

Click the Devices icon. Right-click the display window and select **New > OPC Device**. Fill in the **Group**, **Name**, **Device Type**, **Site** (select the OPC site you previously set up), and **Time Zone**. Configure the other fields as you require.

NOTE: For the **Time Zone** field, select the timezone you want the device data to be displayed in, in the software. This setting is only used for the display of timestamped data in the software. It does not affect the configuration of the monitoring device itself.

Adding a modem

1. Click the **Dial Out Modems** icon, right-click the display window and select **New**, then select either **Serial Modem** (hardware modem) or **WinModem** (Windows modem).

2. Fill in the values for the mandatory fields:

   - For **Serial modem**, select the Modem Type from the list. If your modem is not in this list, select "Generic ION Modem - Other". Select the **Serial Port** where your modem is connected.

   - For **WinModems**, select the Modem Name from the list.

   Configure the other fields as you require.

3. Click **OK**.

WinModem installation notes

Before you can use a WinModem, you must first install it on your computer (follow the instructions that came with the WinModem product). After you restart the computer, set up the WinModem:

1. In Microsoft Windows, click **Start > Settings > Control Panel**, then double-click **Phone and Modem Options**.

2. Click the Modems tab, select your WinModem, then click **Properties**.

3. Click the Advanced tab, then click **Change Default Preference**.

4. Set **Port speed** to match the baud rate that is used between the meter and the modem on the remote end of the connection (both modems must be set to the same baud rate).

5. Set **Flow Control** to "None".

6. Click **OK** to save your changes.

NOTE: You may need to restart your computer for the settings to take effect.

Editing a network component

1. Click the icon in the System Setup pane for the type of component you want to edit.

2. In the display window, right-click the network component you want to edit and select **Configure <Network Component>** (with <Network Component> being the type of component you want to edit).

   You can rename devices or groups displayed in the Devices pane without selecting the **Configure** option.

   > **NOTE**: If you rename a group or device, the data that is already stored in the database is associated with the new group or device name. New data will be added to the database using the new name.

3. Make the required changes. To display all configurable fields, right-click the **Configuration** dialog and select **Advanced Properties**.

4. Click **OK** when you are finished to save your changes.

Deleting a network component

1. Click the System Setup icon for the type of component you want to delete.

2. In the display window, right-click the network component you want to delete and select **Delete** (or select the item and press the DELETE key).

3. Click **Yes** to confirm the deletion or **No** to cancel.

> **NOTE**: When you delete a Site or a Server, all devices associated with it are also deleted.

Renaming devices and groups

You can change the display names of devices and groups without losing any of the real-time or historical data associated with them.

In the Devices pane in Management Console:

- **Group** and **Name** are separate columns.
    - The **Name** column is always displayed.
    - The **Group** column is displayed by default, but you can use the **Column Selector** to remove it from the display area. You can now drag and drop the **Group** column to the **Group By Box** area to show the devices organized within each group.

- Use the **Data Availability** column for inclusion in the display area.
    - **Data Availability** indicates **All** for devices that are active and have real-time and historical data, and **Historical only** for devices that have been deleted and only provide historical data. Historical devices are included in the devices displayed when you select the **Show historical devices** checkbox.

- The **Show historical devices** option displays devices that have been deleted. Since the data for these devices still exists in the system, they can be displayed when you select this option.

Renaming a device

Complete the following steps to change the display name of a device. The system (internal) name is not affected by this name change.

1. Right-click the device name in the Devices pane of Management Console and select **Rename** in the menu to open the **Rename Device** dialog.

2. Enter the new name for the device.

   The appearance of the exclamation icon to the right of the field indicates that there are instructions or errors associated with the device name. Hover over the icon to display the text and take appropriate action.

3. Click **OK** to rename the device.

Changing a group name

Change a group name as follows:

1. Right-click a single group name or select multiple group names (Ctrl + left mouse click) in the Devices pane of Management Console and select **Change Group** in the menu to open the **Change Group** dialog. (You can also select the devices and click the **Change Group** option in the **Edit** menu.)

2. Enter a new group name or select a group name from the list of group names currently used in the system.

   The appearance of the exclamation icon to the right of the field indicates that there are instructions or errors associated with the group name. Hover over the icon to display the text and take appropriate action.

3. Click **OK** to change the group name.

Considerations when renaming devices or groups

Note the following when you rename a device or a group:

- Complete any configuration tasks in progress, such as working in Designer, before changing names.

- Restart any open components such as Vista, Designer, or Web Applications after the name or group change has been completed to refresh the names that appear in them.

- Regenerate network diagrams in Vista to update the top-level grouping objects with the changed Group name.

- Update OPC data mapping after device renaming by selecting the **Update OPC Server** option in Management Console's **Tools > System** menu. You can then update the OPC clients with the new data.

- System (internal) names, not display names, continue to appear in System Log Events in Management Console. You can view the system name of a device, in Device Manager, by selecting **Unique System ID** in the **Column Selector**.

Accessing meters with security

If you have a device in your network with security enabled, use Management Console to allow WinPM.Net Windows Services to access the device. This is required for automatic upload of historical data logs from the device.

To configure WinPM.Net Windows Services to the device:

1. Right-click the device that has security enabled and select **Security** to open the **Meter Security Settings** dialog.

2. From the Software Access drop down list, select the option as per the following table:

Option	Description
Standard Security	Select this option if standard security is configured on the device. **NOTE**: If standard security is not configured on the device, refer to Changes in security settings of device for more information.
User List	Select the user if advanced security is configured on the device.

> **NOTE**: You can configure the device security settings using Designer or ION Setup. See Configuring Security module for more information.

3. Click **Match Password** and type the password in the Password field then re-type it in the **Confirm Password** field.

4. Select the check box if you want to allow the software to send secure time synchronization signals to the device. Clear the check box if you do not want to send secure time synchronization signals to the device.

5. Click **OK**.

Configuring connection schedules

You can schedule the times when the server connects to a site or Ethernet device.

Adding a connection schedule

Click the Connection Schedules icon, right-click the display window, then select **New > Connection Schedule** to open the **Scheduled Site Connection** dialog.

The following sections describe the available tabs and how to configure them.

General Settings tab

On the **General Settings** tab:

- Type the name of your connection schedule (for example "Daily Reading") in the Name field.

- Type a description of the connection schedule, if required.

- Select **Enabled** if you want to enable the schedule.

- Select the date from which the schedule is valid.

Sites tab

On the **Sites** tab:

- Click the box beside the sites and devices to select those that you want to include in the connection schedule. A check mark appears beside an item to indicate that it is included in the connection schedule.

- Set your Site Connection Settings. To enter the minimum and maximum time to stay connected, type the time in the fields or use the arrows to change the time.

- Set the Connection Priority. The higher the number, the greater the priority given to that schedule if there are concurrent connection requests.

Schedule tab

On the **Schedule** tab:

- Set up the date schedule for the connection (the default is Daily). Schedule options vary depending on the type of schedule selected.

  - **Daily**: Occurs every day.

  - **Weekly**: Occurs on the selected day at the specified frequency (in number of weeks) and stays active for the specified duration. For example, if you set Frequency to 2, Day of week to Sunday and Active Duration to 2, the connection occurs every second Sunday and stays active for two days.

  - **Monthly - Day of Week**: Occurs at the specified frequency (in months), in the week and on the day specified and for the specified duration. For example, if you set Frequency to 3, Day of Month to 2nd and Sunday, and Duration to 1, the connection occurs every third month on the second Sunday and stays active for 1 day.

  - **Monthly - Day of Month**: Occurs at the specified frequency (in months), on the specified date of the month, and for the duration specified. For example, if you set Frequency to 2, Day of Month to 15th and Duration to 2, the connection occurs on the fifteenth day of every second month and stays active for 2 days.

- Set up the time period for the connection to occur by setting a start and an end time. In the Start Time and End Time fields, type the time or use the arrows to change the time.

If you configure a full-day (i.e., 24-hour) daily schedule, you need to allow enough time between the end time and the start of the next connection to avoid connection problems. For example, to add a 2-minute gap to a 24-hour schedule, set the Start Time to 0:00:00 and the EndTime to 23:58:00.

System Log events

System Log Events lists events for components in the system. This information comes from the ION_SystemLog database.

> **TIP**: For instructions on filtering and sorting columns and customizing the display, see [Customizing and navigating interface displays](#).

Viewing System Log events

To view System Log Events:

1. Select System Log Events from the System Setup pane.

2. In the **Show** area select the checkboxes for the system components for which you want to view events (i.e., servers, sites, devices).

   If there are no events for a particular system component, that check box is unavailable.

3. Select the minimum priority level of events to be displayed from the **Minimum Priority Level** dropdown list (see below for more information on priority levels).

4. Click **Modify** to change the date/time filter applied to the System Events Log display. See System Log filter setup.

5. To view details of an event, double-click on the entry in the event log.

Priority

Priority classifies what type of event has occurred. The value corresponds to the severity of the event: the higher the number, the more severe the event.

The priority is contained in the ION_SystemLog database as a number. In the System Events Log display, the priority levels are displayed as text strings. The priority numbers are mapped to the text strings as follows:

Diagnostic	0 - 5
Information	6 - 20
Warning	21 - 63
Error	64 - 191
Critical	192 - 255

By default, the *Cutoff* setup register of Log Inserter's System Log Controller module is set to 192. This means that system events with a priority less than 192 are not transferred to the ION_Data database and therefore cannot be viewed in Alarms. If you want to view lower priority events in Alarms, use the above priority mapping as a guide to set the Cutoff setup register to a lower value (using Designer).

System Log filter setup

The System Log Filter Setup dialog lets you change the date and time filter applied to the System Events Log display.

1. Click **Modify** to display the **System Log Filter Setup** dialog.

2. Select the time range by which you want to filter from the dropdown list. To set up a custom filter, select **Custom Date Range** then use the dropdown lists and arrows in the **Select a Date Range** dialog to specify the start and end of the range.

3. Click **OK**.

Setting up large systems

To perform batch network setup of devices, create the first device in Management Console (see Adding network components) using a logical naming scheme. After you create this first device, you can then use it to create multiple duplicate devices and configure their properties.

The Duplicate and Configure interface

> **NOTE**: The Duplicate and Configure function always operates on all devices visible in the table. When table rows are filtered in the Duplicate and Configure tool, changes done using the configuration editors on the top pane of the window affect only the devices that are visible in the Table Editing area. If you want to apply the changes to all devices, clear the filter first before making the changes.

> **NOTE**: The Duplicate and Configure function cannot be used with logical devices. See Creating multiple logical devices for information about creating more than one logical device.

The Duplicate and Configure area

The Duplicate and Configure area has different sections depending on the type of device you are duplicating.

Ethernet devices have the following sections:

- **Configure Names**: Use this section to create the duplicates.

- **Configure TCP/IP Addresses**: Use this section to create a sequence of IP addresses to assign to the duplicates after they have been created.

- **Configure TCP/IP Ports**: The port number used in the original device is automatically assigned to all the duplicates. If required, use this section to select a different port number for all the displayed devices in the Table Editing area (changes made in this area are applied to all devices, including the original).

- **Configure Unit IDs**: This section is disabled by default, since unit IDs do not apply to typical Ethernet devices. (To enable this section, you must first set up the Table Editing area so that the "Unit ID" column is visible).

Serial devices have the following sections:

- **Configure Names**: Use this section to create the duplicates.

- **Configure Unit IDs**: Use this section to create a sequence of unit IDs to assign to the duplicates after they have been created.

- **Configure Serial Sites**: The serial site used in the original device is automatically assigned to all the duplicates. If required, use this section to select a different serial site for all the displayed devices in the Table Editing area (changes made in this area are applied to all devices, including the original).

OPC devices have the following sections:

- **Configure Names**: Use this section to create the duplicates.

- **Configure OPC Address**: Use this section to create a sequence of addresses to assign to the duplicates after they have been created.

- **Configure OPC Sites**: The OPC site used in the original device is automatically assigned to all the duplicates. If required, use this section to select a different OPC site for all the displayed devices in the Table Editing area (changes made in this area are applied to all devices, including the original).

The Table Editing area

When making configuration changes in the Duplicate and Configure interface, these changes are applied only to the rows that are visible in the Table Editing area.

See Customizing and navigating interface displays for instructions on filtering and sorting data in the table.

NOTE: To discard your changes since the last saved state, click Revert. You can also press CTRL+Z to undo the last action. To redo the last action, press SHIFT+CTRL+Z. You can only undo your last action (one undo).

Creating multiple copies of a device

This section describes how to duplicate and configure devices.

Step 1: Configuring the first device

1. Add the first device using your logical naming scheme.

    For example, you can use a scheme like "FL1_PNL1A.CCT01" to name a device that is installed on the "1st Floor" of the building, "electrical panel 1A", and "Circuit breaker #1". Type "FL1_PNL1A" as the group name and "CCT01" as the device name when you configure the first device.

2. Right-click the device you want to duplicate and select **Duplicate and Configure**. The Duplicate and Configure interface appears.

Step 2: Configuring names

In the Configure Names area, the device name is displayed. Each letter and number is selectable. A group of numbers is treated as a single entity. For example, in the name "MY8600" the entities you can select are "M", "Y" and "8600". Leading zeroes are also supported, so you can set sequences such as "01, 02, 03" or "001, 002, 003".

1. Select a letter or number in the device name. The **Start** box displays the selected number/letter, which is the start value. Use the up or down arrow in the spin box to change the start value, or simply type it in the box.

2. Click the up or down arrow on the **End** box edit the range, or type the end value in the box. To clear the range and start over, click **Remove Parameters**.

3. For letters, select Upper Case or Lower Case, depending on whether you require upper case or lower case letters.

4. Set exceptions for individual values or a range of values, if required. Exceptions reduce the number of names created (i.e., total generated names minus the exceptions). See Setting exceptions to generated sequences.

Once you have finished configuring names, you can duplicate the device. See Step 3: Duplicating the device, below, for more information.

Example

For example, if you select the first "1" from the example naming scheme and set the End spin box to "2", two names are generated (the original, plus one duplicate). If you select a letter and set its Start value to "A" and the End value to "C", three names are generated (the original, plus two duplicates).

The total number of generated names is the product of the first range multiplied by the next, and so on until the last range, as indicated in the following graphic.

A	The total generated names are the product: [2 X 3 X 10] = 60

Setting exceptions to generated sequences

The **Exceptions** option applies to the **Configure Names** and **Configure TCP/IP Addresses** sections only.

1. To set exceptions to the range of generated values, click **Edit** (this button is unavailable until you change the name or IP value).
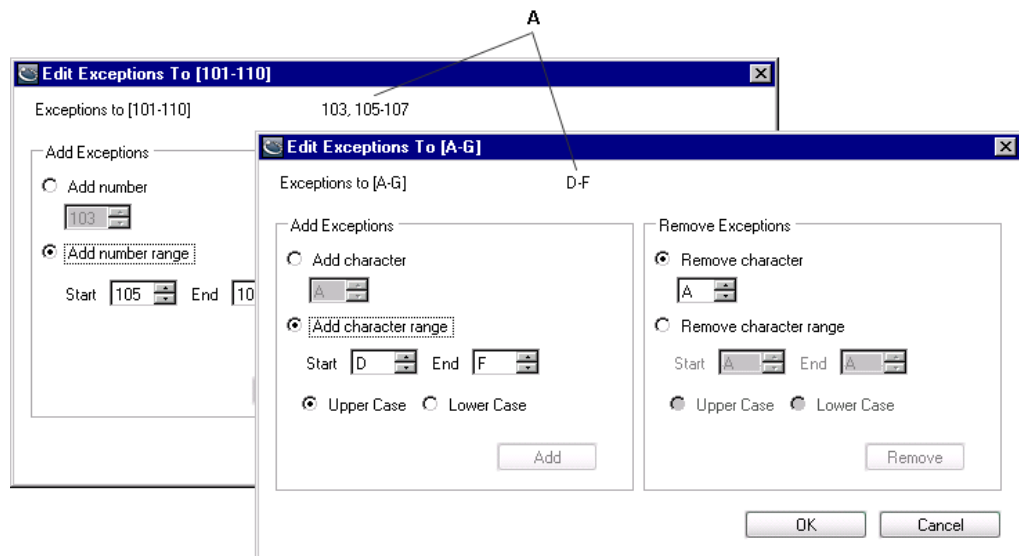
A   List of current exceptions

2. Use the **Add number** or **Add character** (as applicable) spin box to set a value to exclude from the range you have specified, then click **Add**. Repeat to set additional values to exclude. To set a range of values to exclude, select **Add number range** or **Add character range**, then use the **Start** and **End** spin boxes to set the range to exclude.

    Click **Add**. Repeat to set additional ranges of values to exclude.

    NOTE: You cannot add the same number or letter to the list of exceptions more than once. If the **Add** button is disabled, check that the numbers or letters are not already included in the exceptions list near the top of the window.

3. Use the **Remove number** or **Remove character** spin box to remove the exception that was set for a given value (i.e., put it back in as an acceptable value). Use the **Remove number range** or **Remove character range** spin boxes to remove the exception that was set for a range of values. Click **Remove**.

Step 3: Duplicating the device

Once you have finished configuring names, you are ready to duplicate the device. Click **Duplicate**. The duplicated devices appear in the Table Editing area of the window.

The first device in the list is the original device. The remaining devices are the duplicates. Asterisks next to the device names indicate pending changes that have not yet been saved.

Click **Apply** to save your changes and continue editing devices, click **OK** to save your changes and return to Management Console, or click **Revert** to undo unsaved changes.

In Management Console, duplicated devices are always disabled when they are added.

Step 4: Configuring duplicated devices

See the following section, Configuring multiple devices, for instructions on configuring other properties for multiple devices.

Configuring multiple devices

After you have duplicated the devices, you can select and configure any or all of them.

1. If you have closed the Duplicate and Configure interface, select the devices you want to configure from the Devices display window in Management Console. Right-click and select **Configure Selected Devices** (this option is only available if multiple devices are selected).

   The Configure Selected Devices interface appears. This interface has the same layout and controls as the Duplicate and Configure interface; see The Duplicate and Configure interface for information.

   NOTE: Devices affected by a configuration change (that has not been saved) are identified by an asterisk in the first column of the Table Editing area.

2. Scroll to the right in the Duplicate and Configure area to view the Configure TCP/IP Addresses, Unit IDs, Ports and Sites sections of the interface (depending on the type of device being configured). If the number of items in this area does not match the number of items in the |Table Editing area, a message in red text appears in the area. If the numbers match, the message text is black. Configure these parameters as described in Configuring addresses, ports, unit IDs and sites then click **Configure** at the bottom of the applicable interface section.

   NOTE: Configuration changes only affect the devices that are currently displayed in the Table Editing area. Use the column sorting and filtering features, described in Customizing and navigating interface displays, to control which devices are displayed in the table. If you want to apply the changes to all devices, clear any filtering before making the changes.

3. Click **Apply** to save your changes and continue working or click **OK** to save your changes and return to Management Console.

   NOTE: If you changed the name of one or more devices, a message displays with a list of devices that will be renamed. If data for those devices already exist in the ION_Data database and you still proceed with the renaming, then the existing data associated with the old device name is lost ("orphaned").

   Click **Continue** to proceed or **Cancel** to go back and edit the device configuration.

4. When you have finished, click **OK** to save your changes and return to Management Console or click **Cancel** to discard your changes.

   If you click Cancel, a message prompts you if there are pending (unsaved) changes when you click Cancel. Click **Yes** to discard the changes and return to Management Console, or **No** to return to the configuration window and continue editing.

   If inappropriate values (such as a device that is renamed with the same name as another existing device) are encountered, a message displays with error details. You cannot save the configuration changes until you have corrected the errors.

Configuring addresses, ports, unit IDs and sites

The following sections describe how to configure addresses, ports, and unit IDs.

Configuring TCP/IP addresses

1. Select the section of the IP address that you want to configure. The **Start** box displays the selected number/letter, which is the start value. Use the up or down arrow in the spin box to change the start value, or simply type it in the box.

2. Click the up or down arrow on the **End** box edit the range, or type the end value in the box. To clear the range and start over, click **Remove Parameters**.

3. Set exceptions for individual values or a range of values, if required. Exceptions reduce the number of addresses created (i.e., total generated addresses minus the exceptions). See Setting exceptions to generated sequences.

4. Click **Configure** at the bottom of the Configure TCP/IP Addresses section.

Configuring unit IDs

The following instructions apply to serial devices (and Ethernet devices that require a unit ID).

1. Use the up or down arrow in the **Start** spin box to change the start value, or simply type it in the box.

2. Click the up or down arrow on the **End** box edit the range, or type the end value in the box.

3. Click **Configure** at the bottom of the Configure Unit IDs section.

> **NOTE**: To edit the unit ID of an Ethernet device, you must first add the Unit ID column to the table in the Table Editing area.

Configuring sites

1. From the dropdown list, select the site you want to use for the selected serial or OPC devices.

2. Click **Configure** at the bottom of the Configure Sites section.

Configuring ports

1. From the dropdown list, select the port you want to use for the selected Ethernet devices.

2. Click **Configure** at the bottom of the Configure Ports section.

Editing duplicated device information

The following sections describe how to quickly edit the properties of duplicated devices in the table editing area.

Copying device information from a spreadsheet

Sometimes device names do not follow any particular naming pattern. In this case, you can use the copy/paste function to apply configuration changes directly to the Table Editing area.

First, you need to add a new device in Management Console, then create as many duplicates of the device as the quantity of items you are pasting.

If you have a Microsoft Excel spreadsheet that contains a list of these devices to add to Management Console, you can use the Configure Names area to create duplicates of that device type, then use the Table Editing area to copy and paste the device properties from the Excel spreadsheet.

Example: Copy and paste 20 Ethernet devices from excel

1. Confirm that the names in the Excel spreadsheet conform to the naming convention "[Group].[DeviceName]".

2. Add the first device and set up its properties in Management Console. You can assign it a simple device name such as "A.B1" since it will be overwritten when the values are copied and pasted from Excel. Set the Enabled property to No so that Management Console does not attempt to connect to the device.

3. Click **OK** to add the device.

4. Right-click the device and select **Duplicate and Configure**.

5. Select the "1" in the Configure Names box, then use the End spin box to set the value to 20.

6. In the Hide/Show Columns area, clear the check box of all non-editable columns to hide those that contain grayed out values. Also hide the columns you do not want to paste over or those that are not included in the Excel spreadsheet.

7. On the Excel spreadsheet, arrange the columns so they match the order of the columns on the Table Editing area. Hide all columns on the Excel spreadsheet except those corresponding to columns on the Table Editing area.

8. Select the cells in Excel and copy them (CTRL+C).

9. On the Table Editing area, click on the first cell under the Name column to select it, then paste the contents of the clipboard (CTRL+V).

10. Click **Apply** to save your changes or click **OK** save your changes and return to Management Console.

Using Find and Replace to edit device information

Use the **Find and Replace** feature to modify matching characters in the device names or in any other editable columns on the device table. The Find and Replace feature does not work on read-only columns.

1. Right-click the device table and select **Find and Replace** (or press CTRL+H).

2. Type the characters you want to search for in the **Find** box. Do not use wildcard characters ("*"). Note that the search is case-sensitive.

3. **Find in column** lists all available columns. Select the column you want to search in.

4. In the **Replace** box, type the characters you want to use to replace the characters in the **Find** box. If you want to delete characters from a column value, leave the **Replace** box blank.

5. Click **Preview** to list the search results in the table. The **Replace with** column displays the new name or value resulting from the Find and Replace operation.

6. Use the check box in the **Select** column to select the rows for which you want to change values. Click **Select All** to select all the items or **Clear Selected** to clear all.

7. Click **Apply** to save your changes or **Cancel** to discard your changes

8. Click **OK** to save your changes and close the dialog.

Chapter 3: Management Console tools

Management Console tools are additional programs that help you manage and monitor your WinPM.Net network.

(Software) Alarm Configuration

The Alarm Configuration application allows you to configure software alarms for multiple sources and measurements. Each alarm configuration is represented as an Alarm Rule. The rule includes a threshold value, pickup and dropout delays, an update rate, and the alarm priority.

Starting Alarm Configuration

Open the Alarm Configuration application, either from **Tools > Web Tools** menu in Management Console, or **SETTINGS > Configuration Tools** in the Web Applications component.

See *Alarm Configuration Help*, available from the Alarm Configuration application, for information about creating, configuring, and enabling alarm rules.

Database Manager and database maintenance

Database maintenance

WinPM.Net uses databases to store information such as system configuration, data logs, and system event log messages. These databases must be maintained to preserve performance, manage disk space use, and guard against data loss in case of database failure.

NOTICE

LOSS OF DATA

- Back up the database at regular intervals.
- Back up the database before upgrading or migrating the system.
- Back up the database before trimming it.
- Back up the database before making manual database edits.
- Verify correct database behavior after making database or system changes.

Failure to follow these instructions can result in permanent loss of data.

The following table shows the WinPM.Net databases and the recommended database maintenance tasks for each:

Database	Type of Data	Maintenance Tasks*
ApplicationModules	Web Applications related configuration data and system event log entries.	Backup, Maintenance, Trim
ION_Data	Historical power system data such as interval data logs, waveforms and alarms.	Archive, Backup, Maintenance, Size Notification**

Database	Type of Data	Maintenance Tasks*
ION_Network	Device network and other system configuration data	Backup, Maintenance
ION_SystemLog	Non-Web Applications related system event log entries.	Maintenance, Trim

* See Database maintenance task definitions for basic task definitions.

** Size Notification is only used for systems with SQL Server Express, which has a maximum database size limit of 10 GB.

In Standalone WinPM.Net systems, the database maintenance tasks are pre-configured and scheduled to run automatically by default. For Distributed Database WinPM.Net systems, you need to configure the tasks and set up the schedules manually.

NOTE: It is best to automate the maintenance tasks, but you can run them manually on demand using Database Manager and Microsoft SQL Server Management Studio.

The following table shows the default database maintenance task schedules for Standalone WinPM.Net systems:

Database	Task	Enabled	Trigger Time
ApplicationModules	Backup	Yes	Daily at 01:30
ApplicationModules	Maintenance	Yes	Daily at 03:30
ApplicationModules	Trim	Yes	Daily at 02:30
ION_Data	Archive*	**No****	Annually, on Jan 3 at 01:00
ION_Data	Backup	Yes	Weekly, Fridays at 00:00
ION_Data	Maintenance	Yes	Daily at 02:00
ION_Data	Size Notification***	Yes	Daily at 03:00
ION_Network	Backup	Yes	Daily at 01:00
ION_Network	Maintenance	Yes	Daily at 07:30
ION_SystemLog	Maintenance	Yes	Daily at 07:05
ION_SystemLog	Trim	Yes	Daily at 04:00

* The WinPM.Net archive task does not trim the database, it only copies data to the archive.

** You need to edit the Windows user account settings before enabling the archive task. See the Note on the ION_Data archive task for more details.

*** Size Notification is only used for systems with SQL Server Express, which has a maximum database size limit of 10 GB.

For more information on the default task settings see Default maintenance task settings.

Managing database maintenance tasks for Standalone WinPM.Net systems

In Standalone systems, the database maintenance tasks are pre-configured and scheduled to run automatically by default. The scheduling and execution of the tasks is done with Task Scheduler in Windows. The database interaction specific steps of the tasks are defined as Windows

PowerShell scripts.

Note on the ION_Data archive task:

> **NOTE**: The archive task for the ION_Data database is disabled by default. The Windows user account that is used to run this task must have a sysadmin server role in the SQL Server database server. The Windows user account that is used by default, IONMaintenance, does not have a sysadmin server role. To enable and run the scheduled archive task successfully, you need to add the sysadmin role to IONMaintenance, or change the user account that is used to run this task to an account with sysadmin role. See Database maintenance account requirements for more information on account requirements.

To edit the task schedule settings (enable or disable tasks, set trigger times):

> **NOTE**: The database maintenance tasks in Task Scheduler are configured to run using the **IONMaintenance** Windows user account. To save any changes to the task settings in Task Scheduler, you need to enter the password for the **IONMaintenance** account. See Using IONMaintenance for database maintenance tasks for information on where to find the password.

1. On the WinPM.Net application server, open Task Scheduler in Windows.

2. In the Task Scheduler Library, open the **Siemens Industry** > **WinPM.Net** folder to see the configured database maintenance tasks.

3. Edit the tasks as required:

   a. To enable or disable a task, select it and use the **Actions** pane in Task Scheduler.

   b. To edit task settings, double-click a task and make the desired changes in the **Job Properties** dialog box.

4. (Optional) Select **Enable All Tasks History** in the **Actions** pane in Task Scheduler. This turns on event recording for the scheduled tasks, which is useful for auditing and troubleshooting.

5. Close Task Scheduler.

To edit the task script settings (change backup and archive location, set the data to keep on trim, size notification threshold):

1. On the WinPM.Net application server, open the
   `...\WinPM.Net\config\cfg\DbScheduledTasks\Support\Configuration.ps1`
   script file in a text editor.

2. Change the values of the variables in the script file for the settings you want to change. The following settings can be customized:

   > **NOTE**: Follow the instructions in the script file on formatting and syntax.

Setting	Variable	Default Value	Comments
Backup folder location	$locationForBackupFiles	...\ WinPM.Net \Database\Backup	Sets the folder to which the database backups are saved. The backup script will create the following subfolders in this location: ...\Data for ION_Data ...\Network for ION_Network ...\SystemLog for ION_SystemLog ...\Applications for ApplicationModules NOTE: IONMaintenance needs Read and Write permissions on this folder.
Archive folder location	$locationForArchiveDBFiles	...\ WinPM.Net \Database\Archive	Sets the folder to which the database archives are saved. NOTE: IONMaintenance needs Read and Write permissions on this folder.
Data to keep when trimming	$diagnosticsDaysToKeep	30	Sets how many days' worth of data is left in the database after trimming.
Database size (max)	$maximumDatabaseSizeIn Gigabytes	9	Sets the maximum database size. This value is used by the size notification task to assess what percentage of database space has been used. The maximum size for a SQL Express database is 10GB. The maximum size in the script is set to 9GB to allow for a 1GB warning buffer before the database stops logging data.
Database size notification limit	$databaseSizeNotification ThresholdPercentage	85	Sets the threshold for when a database size notification will be issued.

3. Save the script file changes and close the text editor.

Database Manager

Use Database Manager to manually perform operations on the WinPM.Net databases.

NOTICE

LOSS OF DATA

- Back up the database at regular intervals.
- Back up the database before upgrading or migrating the system.
- Back up the database before trimming it.
- Back up the database before making manual database edits.
- Verify correct database behavior after making database or system changes.

Failure to follow these instructions can result in permanent loss of data.

NOTE: Database Manager does not include manual database operations on the Application Modules database (ApplicationModules). In a disaster recovery situation or when directed by Technical Support, the Application Modules database can be restored by using the Restore database function in the SQL Server Management Studio. See Restoring a database for further information about this operation. WinPM.Net databases can be restored using the same process.

For information on database maintenance tasks in WinPM.Net, see Database maintenance.

Prerequisites

The following user prerequisites need to be met to work with Database Manager and database functions through SQL Server Management Studio:

- For Database Manager: Since Windows authentication is used to access the SQL Server databases, the user needs to be a member of the sysadmin SQL Server role, which is set in SQL Server Management Studio.

- For most database functions available in SQL Server Management Studio: The user needs to be a member of the sysadmin SQL Server role.

NOTE: If the Windows user that you used to log into the system is not a member of the sysadmin role, and you want to run Database Manager, you can do so without logging out by completing the following steps:

1. Navigate to the system\bin folder in the product's install location.
2. Locate DatabaseManager.exe.
3. Click the EXE file name to highlight it, then press **Shift+Right-click** to open the menu.
4. Click **Run as different user** to open the Windows Security dialog.
5. In the **User name** field, type a user name that has the sysadmin role, then type the password for that user.
6. Click **OK** to open Database Manager.

Database Manager interface

To access Database Manager, start Management Console then click **Tools > Database Manager**.

Note that the Windows user running Database Manager needs to be a member of the sysadmin SQL Server role.

Expand the items in the navigation tree to display the **Databases** for each instance.

Note that scheduled jobs are available only in Windows Task Scheduler. For further information, see Database maintenance.

A	SQL Server Instance	B	Databases	C	Options menu	D	Show Archives
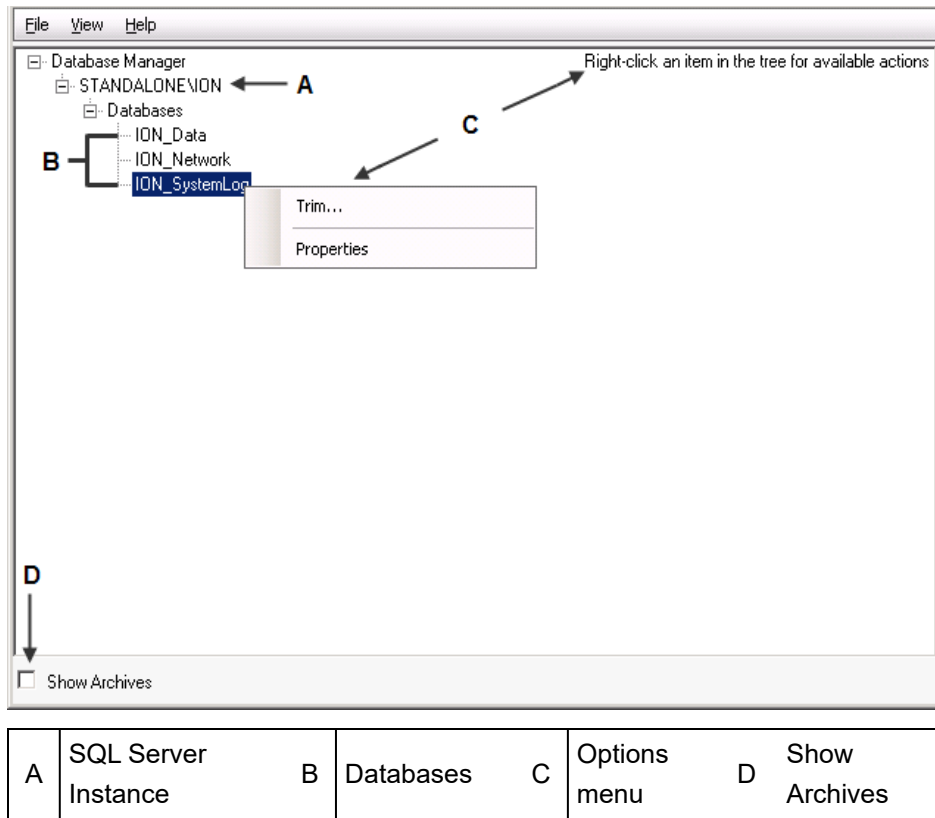
SQL Server instance

The default SQL Server instance that the product uses for its databases is *COMPUTERNAME \ INSTANCE*, where *COMPUTERNAME* is the name of the server, and *INSTANCE* is the SQL Server instance used with the product.

Databases

The **Databases** section lets you view information about the databases or perform manual actions on the databases.

For information on the manual actions you can perform, see Manual actions.

Viewing Database Properties

To view the properties of a particular database, right-click that database and select **Properties**. The properties are:

- **Size**: The current size of the database.

- **Primary File Location**: The file path for the primary database (.mdf) file.

- **Transaction Log Location**: The file path for the transaction log (.ldf) file.

- **Creation Date**: The date and time when the database was created.

- **Last Backup Date**: The date when the last backup was performed.

- **Disk Space Available**: The amount of free space available on the disk where the database resides.

- **Server Version**: The type and version of the SQL Server instance that is hosting the database.

Show archives

Select this check box (lower left-hand corner) if you want the list under **Databases** to include all archived databases along with the live databases. After **Show Archives** is selected, you can upgrade archived databases or view the properties of the archived databases.

Clear the **Show Archives** check box to hide the archived databases from view. This also prevents the database actions from being performed on database archives.

Manual actions

The following sections provide information on the manual actions that you can use to manage your databases.

To perform an action manually, do one of the following:

- Right-click **Databases** and select the action from the menu, or
- Right-click the specific database and select the action you want to perform from the pop-up menu.

When you right-click **Databases** and select an action, a dialog specific to that action opens. The databases listed in the dialog are those to which the action applies.

When you right-click a specific database, only the actions that apply to that database appear in the menu and the database is selected by default in the dialog for the action.

Archive

The Archive action creates an archive of the selected database.

Before proceeding, ensure that you have write access to the archive directory location.

1. Right-click **Databases** or **ION_Data** and select **Archive** to open the **Database Archive** dialog. If necessary, select the database that you want to archive.

2. Select the directory where the archive will be saved.

   For **standalone** environments:

   a. In **Save archive to**, click the browse button ⬚ to select the directory where the archive will be saved.

   `...\WinPM.Net\Database\Archives\Data` is the default directory for the saved archive, but you can specify another local directory.

   b. (Optional) Specify a different local directory.

   > **NOTE**: You can only save an archive to a directory on the local machine, not to a location on the network.

   c. (Optional) Edit the default archive filename to follow your naming conventions.

   > **NOTE**: The database name is restricted to characters A-Z, a-z, 0-9, and _ (underscore).

For **distributed** environments:

a. In **Save archive to**, enter an existing directory path on the server and a valid filename for the archive.

For example: `C:\Archives\ION_Data_January.mdf`. The path – in the example `C:\Archives` – must exist on the Database Server.

> **NOTE**: The database name is restricted to characters A-Z, a-z, 0-9, and _ (underscore).

3. Select the data types that you want to archive.

4. Specify the date range of the data that you want to archive. For **Start**, select **The beginning of the database** or select **Date** and enter a date and time. Enter a date and time for **End Date**.

5. Under **Trim after archive** select whether or not you want to remove archived data from the database.

   You need to select **The beginning of the database** for the start date range for trimming the live database, otherwise the **Trim after archive** option is disabled.

6. Click **OK**.

The **Progress** field displays the current progress of the archive process. If a manual archive does not succeed, a message appears and the **Database Archive** dialog remains open with the **OK** button grayed out — examine the **Progress** field to discover where the process did not succeed. If the archive is successful, the dialog closes automatically.

Export Registry Setting

The Export Registry Setting action exports settings to a registry (.reg) file. This is useful if you need to set up client computers in a system where the primary server's database settings (server instance name and database name) are customized. After you export the settings to the registry (.reg) file, you can import that registry file on the client computer.

1. Right-click **Databases** and select **Export Registry Setting** to open the **Database Registry Key Export** dialog.

2. Type the path and filename for the file or click the browse button ⬚ to specify the location for the saved the database registry key.

   The directory ...\WinPM.Net\config\cfg\ is the default save location for the exported database registry key.

3. Click **OK**.

New ION_Data Database

The New ION_Data Database action creates a new, blank version of the ION_Data database.

1. Right-click **Databases** and select **New ION_Data Database** to open the **New Historical Database** dialog.

2. Type a name for your new database.

> **NOTE**: Do not name it "ION_Data" as this is the default name for the existing ION database. Database names are restricted to characters A-Z, a-z, 0-9, and _ (underscore).

3. Click the browse button ⬚ to specify a location for the database.

4. Click **OK**.

Trim

The Trim action removes data from a database.

<table>
<tr><td align="center"><h2>NOTICE</h2></td></tr>
<tr><td>

LOSS OF DATA

- Back up the database at regular intervals.
- Back up the database before upgrading or migrating the system.
- Back up the database before trimming it.
- Back up the database before making manual database edits.
- Verify correct database behavior after making database or system changes.

Failure to follow these instructions can result in permanent loss of data.

</td></tr>
</table>

1. Right-click **Databases**, **ION_Data** or **ION_SystemLog** and select **Trim** to open the **Trim Database** dialog.

2. Select the database you want to trim from the **Database to trim** list.

3. Under **Trim Range**, specify the date range of data you want to trim (for the ION_Data database) or set the maximum data age in days (for the ION_SystemLog database).

4. For the ION_Data database, select the data types you want to trim (**Data Records**, **Waveforms**, **Events**) in the **Data Types** section. You can select any combination of data types to trim.

5. Click **OK**. A message appears to notify you that the selected data will be removed. Click **Yes** to continue or **No** to cancel.

Upgrade Database

The Upgrade Databases action upgrades the selected database to the latest database schema.

1. Right-click **Databases** and select **Upgrade Databases** to open the **ION Database Upgrade** dialog.

   The WinPM.Net installer automatically upgrades your databases with the new database schemas when you install WinPM.Net on an existing server. If you install the latest version of the product on a different server so that you can manually copy older database files to the new computer, you can run this action on the older databases (that is, on the ION_Data, ION_SystemLog, and ION_Network databases, and archives) to upgrade them with the new schema.

2. Select the database in the list that you want to upgrade and click **OK**.

Restoring a database

You can restore a database from a backup by logging in to SQL Server Management Studio as a user with syadmin access authority for the **Restore Database** function. (Database backups are specified as a scheduled job in Windows Task Scheduler. See [Database maintenance](#) for more information.)

Restoring the latest database

Complete the following to restore a database from the latest backup:

1. Stop all WinPM.Net services.

2. Open **SQL Server Management Studio**, enter your password if required and click **Connect** to access your SQL Server.

3. In the **Object Explorer** pane on the left, expand **Databases**, right-click the database you want to restore and click **Tasks > Restore > Database** to open the **Restore Database** dialog.

4. Under **Source**, select **Database** and click the database you want to restore in the dropdown list if it is not already selected.

5. Under **Backup sets to restore**, select the checkbox in the **Restore** column for the database you want to restore.

6. Click **Options** in the **Select a page** pane on the left.

7. On the **Options** page:

    – Under **Restore options**, select **Overwrite the existing database (WITH REPLACE)**.

    – For **Recovery state**, select **RESTORE WITH RECOVERY** from the dropdown list.

       RESTORE WITH RECOVERY is described as **Leave the database ready to use by rolling back uncommitted transactions. Additional Transaction logs cannot be restored**.

8. Click **OK** to begin the restore operation.

    A message indicates that the database has been restored successfully. If the restore operation is not successful, the database reverts to its original state.

    NOTE: After you restore the database, you need to assign its ownership to the ION user as follows:

    a. In SQL Server Management Studio, right-click the restored database and click **Properties** in the menu to open the Database Properties dialog.

    b. Click **Files** under **Select a page**.

    c. Click the button on the right of the **Owner** field to open the Select Database Owner dialog.

    d. Type ION in the field labeled **Enter the object names to select** and click **Check Names** to adjust the format of your entry to [ION].

    e. Click **OK** to update the owner of the database.

      f.   Click **OK** to close the Database Properties dialog.

9. Restart all WinPM.Net services.

Restoring a specific database

Complete the following to restore a specific database:

1. Repeat steps 1 through 3 from [Restoring the latest database](#) above.

2. Under **Source**, select **Device** and click **Browse** to open the **Select backup devices** dialog.

3. Select **File** in **Backup media type** list if it is not already specified and then click **Add** to open the **Locate Backup File** dialog.

4. Navigate to and select the backup file you want to restore and click **OK**.

5. Verify that the file referenced in the **Specify Backup** dialog is the one you selected and click **OK** to return to the **Restore Database** dialog.

6. Under **Select the backup sets to restore**, select the checkbox in the **Restore** column for the database you are restoring.

7. Click **Options** in the **Select a page** pane on the left.

8. On the **Options** page:

   – Under **Restore options**, select **Overwrite the existing database (WITH REPLACE)**.

   – Under **Recovery state**, select **RESTORE WITH RECOVERY** from the dropdown list.

   RESTORE WITH RECOVERY is described as **Leave the database ready to use by rolling back uncommitted transactions. Additional Transaction logs cannot be restored**.

9. Click **OK** to begin the restore operation.

   A message indicates that the database has been restored successfully. If the restore operation is not successful, the database reverts to its original state.

   > **NOTE**: After you restore the database, you need to assign its ownership to the ION user as follows:

   a. In SQL Server Management Studio, right-click the restored database and click **Properties** in the menu to open the Database Properties dialog.

   b. Click **Files** under **Select a page**.

   c. Click the button on the right of the **Owner** field to open the Select Database Owner dialog.

   d. Type ION in the field labeled **Enter the object names to select** and click **Check Names**.

   The format of your entry changes to [ION].

   e. Click **OK** to update the owner of the database.

   f. Click **OK** to close the Database Properties dialog.

10. Restart all WinPM.Net services.

Accessing the ION Database with 3rd-party applications

The following example uses Microsoft Excel 2007 to retrieve data from the ION_Data database.

NOTE: The database uses Universal Coordinated Time (UTC) time. When you import data into Excel, the Clock module's Time Zone Offset register for the meter is applied.

1. To create a database query in Excel, click **Data > From Other Sources > From Microsoft Query**.

2. To create a new data source, select <New Data Source> from the list and click **OK**.

3. Type the name you want to give the data source. Once you type a name, the list of drivers for the database type becomes available. Select <SQL Server> from the list. Click **Connect**.

4. Choose your server from the list. Provide your LoginID and Password and click **OK**. Select a default table for your data source (optional). Click **OK** to continue.

5. Select the source name you created and click **OK**.

6. After you have selected the data you want to view, the **Query Wizard** opens. Use this to filter and sort the data.

7. Follow the steps in the **Query Wizard** to add data to the spreadsheet.

Deactivating alarms

Active alarms can continue to appear in the Alarms viewer if, for example, an alarm dropout has not been configured, or a device has been removed from the network after an alarm was issued. In such cases you can use the Deactivate Alarms utility to remove these permanently active alarms from the Alarms viewer.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

To use the Deactivate Alarms utility:

1. Open **Management Console** and select **Tools > Deactivate Alarms** to open the dialog.

2. Select a date for the **Show Active Alarms older than** field if you want to change the default date.

3. Click **Load Active Alarms** to display a list of the active alarms that are older than the date specified.

4. Use the **Select** column to select the alarms that you want to set to an inactive state. You can also click **Select All** to select all of the alarms displayed in the grid. Use Select None to clear the selection of any alarms listed.

5. Click **Set Selected Alarms to Inactive**.

    The **Active** column displays **N** for all alarms set to the inactive state.

6. Click **Done** to close the Deactivate Alarms dialog and then close Management Console.

    When you log in to **Management Console** again and open the **Deactivate Alarms** dialog, the number of alarms shown in the **Alarms Displayed** field matches the number in the **Alarms Displayed** field in the **All Active Alarms** view in the **Alarms** application. (The Alarms application is available in the Web Applications component.)

Device Type Editor

NOTE: The Device Type Editor replaces the Modbus Device Importer (MDI) tool that was available in older versions of the software.

Use Device Type Editor to create device drivers for Modbus and OPC devices, in WinPM.Net, for which pre-configured drivers or downloadable add-on drivers do not exist . You can also edit existing Modbus and OPC device drivers with Device Type Editor.

Device drivers are called Device Types in WinPM.Net.

NOTE: Before creating a new device type with the Device Type Editor, check if it is available as a pre-configured device type or as a downloadable add-on.

NOTE: The device types you can create with the Device Type Editor are limited to reading numeric and Boolean data from the device, and writing simple 16-bit values to the device. These device types cannot access complex data such as historical logs, event logs, or waveform captures.

NOTE: Drivers created with the Device Type Editor require mid-range (M) type device licenses.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software to control time-critical functions.
- Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

Creating a Modbus Device Type

A Modbus device type translates between the register based data structure on the Modbus device and the ION based data structure in WinPM.Net.

When you create the device type with the Device Type Editor, you first define the Modbus registers that you want to access on the device, and then you map these registers to ION managers, modules, and registers. You can enter the Modbus register information manually in the Device Type Editor user interface, or use Microsoft Excel to define the register list and then copy and paste it into the Device Type Editor. The mapping of Modbus to ION registers is done by dragging Modbus registers onto ION registers, or the reverse, using the mouse pointer.

Prerequisites

To create a device type for a device, you need to know the details of its Modbus register map. At a minimum, you need the Modbus addresses, data formats, and possible scaling values for the measurements you want to access on the device. You can get this information from the device documentation or the device manufacturer.

> **NOTE**: Before creating a new device type with the Device Type Editor, check if it is available as a pre-configured device type or as a downloadable add-on.

References

See the following references for information on how to use the Device Type Editor or information related to Modbus register definitions:

- Device Type Editor User Interface
- Configuring Logging and Calculation User Interface
- Modbus Data Formats
- Modbus Register Properties
- Measurement Tree

To create a Modbus device type:

1. In Device Type Editor, click **File > New > Modbus Device Type** to open a New Mapping template.

2. Enter the Modbus register information into the Modbus Map table in the right display pane, in one of two ways:

   - Add each Modbus register manually in the Device Type Editor, using the available editing controls (right-click to open the context menu).

   - Use Microsoft Excel to define the register list, copy it, and paste it into the Modbus Map table.

     > **NOTE**: The register list columns in the Excel sheet must match the column content and positions in the Device Type Editor Modbus Map table. Confirm that the format entries match one of the supported Modbus Data Formats and that there are no spelling mistakes, such as added space characters or different capitalization.

3. Map each Modbus register in the Modbus Map to a measurement in the Measurement Tree in the left pane. To see the measurements in the tree, expand the managers and modules, for example Current > Phase Current > Current A,B,C, ... Map the registers in one of two ways:

   - Using the mouse, drag the measurement you want to map in the Measurement Tree onto the corresponding measurement in the Modbus Map.

   - Using the mouse, drag the Modbus register you want to map in the Modbus Map onto the corresponding measurement in the Measurement Tree.

     > **NOTE**: If you cannot find a suitable measurement in the Measurement Tree, add a new manager, module, and register to represent this measurement. To do this, right-click any of the existing managers in the Measurement Tree and choose the appropriate menu options.

4. Configure the device type name and template name.

   a. In the Measurement Tree, click **Factory Information > Factory 1 > FAC1 Device Type** to open the String Register Details dialog for this register. In String Register Details, enter a device type name in the **Value** box. This name will be visible in the Management Console when you select a device type for a new device. Choose a name that identifies the device, but does not match any of the existing device types.

Example: For a device called "PM123", you could use a device type name of "DTE_PM123". The device type name does not have to include DTE, but it is a good practice to choose a string that identifies this device type as a DTE type.

b. In the Measurement Tree, click **Factory Information > Factory 1 > FAC1 Template** to open the String Register Details dialog for this register. In String Register Details, enter a template name in the **Value** box. This name is used to match your new device type with a default Vista diagram, if you choose to create such a diagram. The template name should include the device type name followed by a version number that represents the version of your new device type.

Example: For a device type called "DTE_PM123", you could use a template name of "DTE_PM123_V1.0.0".

5. (Optional) Click **Tools > Clean Measurement Tree** to remove any Measurement Tree elements that have not been mapped to Modbus registers. This makes it easier to find the relevant measurements in the Measurement Tree in Vista or OPC clients for example.

6. Click **File > Save**, to save the new device type.

Related topics

- Creating a Modbus Device Type

- Editing a Modbus Device Type

- Adding Software Logging to a Modbus Device Type

- Installing a Device Type on the System

Editing a Modbus Device Type

You can use the Device Type Editor to modify existing Modbus device types. This includes your own, as well as factory types, which means pre-configured and downloadable add-on device types. The factory device types have been designed and tested for compatibility with the devices and the WinPM.Net software. If you see a need to modify a factory device type, save the modified type under a new name that does not match any of the factory device type names. This ensures that you can identify this type as custom and it prevents it from being overwritten during a system upgrade.

References

See the following references for information on how to use the Device Type Editor or information related to Modbus register definitions:

- Device Type Editor User Interface

- Configuring Logging and Calculation User Interface

- Modbus Data Formats

- Modbus Register Properties

- Measurement Tree

To edit a Modbus device type:

1. In Device Type Editor, click **File > Open**, find and select the device type you want to edit, and click **Open**.

2. Modify the Modbus Map registers, the Measurement Tree mappings, or the software logging to meet your needs.

   See Creating a Modbus Device Type for details on how to perform these tasks.

3. Click **File > Save**, to save the modified device type under the same name. Click **File > Save As**, to save the modified device type under a new name.

Related topics

- Creating a Modbus Device Type
- Editing a Modbus Device Type
- Adding Software Logging to a Modbus Device Type
- Installing a Device Type on the System

Adding Software Logging to a Modbus Device Type

The Modbus device types you create with the Device Type Editor are limited to reading numeric and Boolean data from the device. These device types cannot access complex data such as historical logs, event logs, or waveform captures. However, it is possible to automatically log the measured values in WinPM.Net after they are read from the device. In addition to logging the measured values, WinPM.Net can also calculate and log average, low, and high values for these measurements.

NOTE: The data logs are timestamped with the date and time at which the software logs the value, not the date and time the measurement was taken on the device. Device communications and system delays determine the time delay between when the measurement is taken by the device and when it is logged by the software.

To add software logging:

1. In Device Type Editor, open the Modbus device type for which you want to add logging.

2. Click **Tools > Configure Software Logging** to open the Configure Logging and Calculation dialog.

3. In Configure Logging and Calculation, select the measurements you want to log and calculate, and set the logging intervals.

   NOTE: Only numeric output registers can be calculated and logged. All other types of registers (for example, enumerated) do not appear on the Configure Logging and Calculation screen.

   See Configuring Logging and Calculation User Interface for information on the controls and options available in the user interface.

4. Click **OK** to save the changes.

Related topics

- [Creating a Modbus Device Type](#)

- [Editing a Modbus Device Type](#)

- Adding Software Logging to a Modbus Device Type

- [Installing a Device Type on the System](#)

Creating an OPC Device Type

An OPC device type translates between the tag based data structure on the OPC server and the ION based data structure in WinPM.Net.

When you create the device type with the Device Type Editor, you first define the OPC tags that you want to access on the server, and then you map these tags to ION managers, modules, and registers. You can enter the OPC tag information manually in the Device Type Editor user interface, or use Microsoft Excel to define the tag list and then copy and paste it into the Device Type Editor. The mapping of OPC tags to ION registers is done by dragging tags onto ION registers, or the reverse, using the mouse pointer.

NOTE: The software logging of values is not available for OPC device types.

Prerequisites

To create a device type for an OPC server, you need to know the details of its tag list. You can get this information from the OPC server documentation or the OPC server vendor. You can also get the tag list by browsing the OPC server with an OPC client such as the OPC Test Client we ship with WinPM.Net. You can find the OPC Test Client, `OpcTestClient.exe`, in
`...\WinPM.Net\system\bin\`

References

See the following references for information on how to use the Device Type Editor or information related to OPC tag definitions:

- [Device Type Editor User Interface](#)

- [Measurement Tree](#)

- [OPC Tags](#)

To create an OPC device type:

1. In Device Type Editor, click **File > New > OPC Device Type** to open a New Mapping template.

2. Enter the OPC tag information into the OPC Map table in the right display pane, in one of two ways:

   - Add each tag manually in the Device Type Editor, using the available editing controls.

   - Use Microsoft Excel to define the tag list, copy it, and paste it into the OPC Map table.

     NOTE: The tag list columns in the Excel sheet must match the column content and positions in the Device Type Editor OPC Map table. .

NOTE: When entering a tag, only specify the tag name, with a leading period (.), do not include the OPC server or device name. For example, for the tag `Opcda://Server/OpcDa.1.Channel_1.Powers.Real_Power` enter `.Powers.Real_Power` as the tag name in the OPC Map.

3. Map each OPC tag in the OPC Map to a measurement in the Measurement Tree in the left pane. To see the measurements in the tree, expand the managers and modules, for example Current > Phase Current > Current A,B,C, ... Map the tags in one of two ways:

   – Using the mouse, drag the measurement you want to map in the Measurement Tree onto the corresponding tag in the OPC Map.

   – Using the mouse, drag the OPC tag you want to map in the Modbus OPC Map onto the corresponding measurement in the Measurement Tree.

NOTE: If you cannot find a suitable measurement in the Measurement Tree, add a new manager, module, and register to represent this measurement. To do this, right-click any of the existing managers in the Measurement Tree and choose the appropriate menu options.

4. Configure the device type name and template name.

   a. In the Measurement Tree, click **Factory Information > Factory 1 > FAC1 Device Type** to open the String Register Details dialog for this register. In String Register Details, enter a device type name in the **Value** box. This name will be visible in the Management Console when you select a device type for a new device. Choose a name that identifies the OPC server, but does not match any of the existing device types.

      Example: Use "OPC_ABC" as the device type name, where ABC identifies the server.

   b. In the Measurement Tree, click **Factory Information > Factory 1 > FAC1 Template** to open the String Register Details dialog for this register. In Edit String Register, enter a template name in the **Value** box. This name is used to match your new device type with a default Vista diagram, if you choose to create such a diagram. The template name should include the device type name followed by a version number that represents the version of your new device type.

      Example: For a device type called "OPC_ABC", you could use a template name of "OPC_ABC_V1.0.0".

5. (Optional) Click **Tools > Clean Measurement Tree** to remove any Measurement Tree elements that have not been mapped to OPC tags. This makes it easier to find the relevant measurements in the Measurement Tree in Vista or an OPC client for example.

6. Click **File > Save**, to save the new device type.

Related topics

- Creating an OPC Device Type

- Editing an OPC Device Type

- Installing a Device Type on the System

Editing an OPC Device Type

You can use the Device Type Editor to modify existing OPC device types.

References

See the following references for information on how to use the Device Type Editor or information related to OPC tag definitions:

* [Device Type Editor User Interface](#)
* [Measurement Tree](#)
* [OPC Tags](#)

To edit an OPC device type:

1. In Device Type Editor, click **File > Open**, find and select the device type you want to edit, and click **Open**.

2. Modify the OPC tags or the Measurement Tree mappings to meet your needs.

    See [Creating an OPC Device Type](#) for details on how to perform these tasks.

3. Click **File > Save**, to save the modified device type under the same name. Click **File > Save As**, to save the modified device type under a new name.

Related topics

* [Creating an OPC Device Type](#)
* Editing an OPC Device Type
* [Installing a Device Type on the System](#)

Installing a Device Type on the System

Before a device type can be used by WinPM.Net to communicate with a device, it must be installed on the system. Follow the steps below to install a new device type or a modified device type.

To install a new or modified device type:

1. In Device Type Editor, click **Tools > Install Device Type**, to open the Device Type Installer.

2. In Device Type Installer, specify the device type you want to install.

    (Optional) Specify a diagram and icon to be used for this device type in the Vista and Diagrams applications if you want to use the default diagram feature.

    (Optional) Specify a measurement extension file for exposing the device measurements through the OPC server, if desired.

3. Check the Device Type Name and Template. If you need to make changes, exit the Device Type Installer, open the device type for editing in the Device Type Editor and correct the names, then run the installer again.

4. Select the **Protocol** for your device type, **Modbus** or **OPC**.

    (Optional) Select **Ethernet Support** if the device has Ethernet communication capabilities.

5. Click **Install Device Type**, to install the device type on the system.

Related topics

- [Creating a Modbus Device Type](#)

- [Editing a Modbus Device Type](#)

- [Adding Software Logging to a Modbus Device Type](#)

- [Creating an OPC Device Type](#)

- [Editing an OPC Device Type](#)

- Installing a Device Type on the System

Diagnostics Viewer

Diagnostics Viewer is a tool you can use to troubleshoot network communications problems and related network errors.

Starting Diagnostics Viewer

1. Start Management Console and log in.

2. Click **Tools > System > Diagnostics Viewer**.

TIP: For instructions on using filtering, sorting, column selection, and pin/unpin to customize the Diagnostics Viewer display, see [Customizing and navigating interface displays](#).

Navigation pane

Diagnostics information is grouped as follows:

- **Service Diagnostics**: Contains diagnostics information for certain services (WinPM.Net Network Router Service, WinPM.Net Site Service and WinPM.Net Log Inserter Service).

- **Communication Diagnostics**: Contains diagnostics information for the sites, hardware devices, and software nodes.

Select an item in the navigation pane to display its diagnostics information.

If you add a new device to the system while Diagnostics Viewer is open, you can refresh the tree view to display the new device by collapsing then expanding the root node of the tree.

Diagnostics Information pane

The diagnostics information pane displays detailed data about the state of your power monitoring system and devices.

Service Diagnostics

Service Diagnostics records communication problems and similar events occurring with the product's software components.

Communication Server diagnostics

Information about the communications server is arranged in these tabs:

- **Console Messages** lists all WinPM.Net Network Router Service and WinPM.Net Site Service console messages for the current session.

> **TIP**: The blank area below the **Description** column header is a dynamic filter field. Type the wildcard character (*) in front of the text you want to search (for example, to display only messages prefixed with WARNING, type ***warning**). The diagnostics information pane automatically displays only those records that match the text you typed in the box.

- **Connection Status** displays the current status of the software components connected to Network Router.

- **Tree States** displays the ION tree status of all nodes (hardware devices and software nodes).

Log Inserter diagnostics

The Log Inserter diagnostics information pane is split into two sections. The top section (Select Nodes pane) contains the available nodes, while the bottom section contains the node details.

Select nodes to display

In the **Select Nodes** pane, select the check box beside a node to display its diagnostics information. Clear the check box to hide that node's diagnostics information.

> **TIP**: If there are many nodes and you want to display only a few of them, right-click the **Select Nodes** area then click **Clear All**. Select only the nodes you want to display. To display all the nodes again, right-click the **Select Nodes** area and click **Select All**.

Node details

The node details are organized in these tabs:

- **Node Information** provides diagnostics associated with the communication status of each selected node. If the Log Inserter is not configured to gather data from a given node, it does not appear in the list in the **Node** column. If the Log Inserter is configured to automatically gather information for a node, but that node has not yet been processed, it does not initially appear in the list. Once information becomes available, the node appears (if it has been selected).

- **Node Performance** provides per-node performance summary information.

- **Log Performance** provides performance information on a per log basis.

The following table summarizes the columns on the **Node Information** tab:

Column	Description
Node	The name of the device, VIP, or Log Inserter.
DeviceType	The device type of the associated node that is returned by the device itself. The Log Inserter uses this to detect device swap outs.
SerialNumber	The serial number of the device that is returned by the device. The Log Inserter uses this to detect device swap outs.
Configured Polling Interval (s)	The requested polling interval in effect. It can be configured either from the log upload control or from the custom Windows Registry value. All of the nodes for which polling is disabled are identified with Polling Disabled in this column.

Column	Description
Average Update Interval (s)	A weighted average time between polled results for the device. The most recent interval accounts for 20% of the value, and the previous average accounts for the remainder. If the value deviates from the average by more than 30s, then the old average is discarded and the current interval is used. By default, the expected value for devices that support logs is the Configured Polling Interval (s) value. The expected value for devices that do not support logs is 60 seconds. If the Log Inserter is selected but it is not configured to collect data from its System Log Controller, it appears in the diagnostics and shows 300s for Average Update Interval. Initially this value is `n/a`.
Time Since Update (s)	The time in seconds since the last communication with the node. This time includes polling updates, record uploads, and configuration loads.

Column	Description
CommStatus	Can be one of the following values: • `alive` – The node is communicating. • `late` – If a response to a polling program is not received within 3 minutes, the Log Inserter sends a ping. If the ping does not respond in 10 minutes, the communication status is set to `late` and another ping is sent. The system continues pinging every 10 minutes until a response is received. • `expired` – If a ping returns before the response for any preceding request, the original request was lost. The request is abandoned and the communication status is set to `expired`. A request can be lost if a destination Site Server or VIP is shut down. The state changes from `expired` when the device responds to a request. • `timeout` – A request to the device timed out. The device is not communicating. • `site not connected` – The site is currently not connected. • `cannot send` – An unrecoverable error. The Log Inserter cannot send a program to the communications subsystem. The Log Inserter shuts down if Network Router is not running. Restart the system. • `invalid password` – The password entered for this device in Management Console is invalid. • `password changed` – The password for this device has been changed. Update the password for the device in Management Console. • `site not responding` – The connection is unexpectedly broken during communication with the device. • `device disabled` – The device or its site is not enabled. Note that Log Inserter automatically removes this node from this list if the node had been detected by automatic means. • `does not exist` – The device is not registered in the system. In auto-mode, the device eventually disappears from the list unless it is referenced remotely by a VIP. • `pending` – No responses have been processed. • `nack'd` – The request was not acknowledged. This could mean that the Site Server hosting the device is not running. • `validating` – Treemon reported that the device is not responding. A signal is sent to Treemon to validate the state. This state clears once Treemon (via Validator) establishes

Column	Description
	communications with the device.
Comments	Under steady-state conditions, this is blank. While the Log Inserter attempts to upload configuration information, this can contain a string value indicating that the `Tree is in use by another client`. This indicates that the Log Inserter cannot process the device until the aforementioned client releases it. If the client is ION Designer, it is not released until the node is closed in Designer or Designer is closed. If the client's name ends with `-not-clean`, the node is currently being evaluated by Treemon/Validator.
AggregateSetupCount	The aggregate setup count of the device. The Log Inserter uses this to detect configuration changes.
RequestedIONs	The number of ION registers, modules, and/or managers that have been requested from the tree. The Log Inserter needs to upload configuration information to determine which logs need to be processed, which labels should be used for measurement mapping and source resolution, and which labels to use for event cause and effects. The Log Inserter retrieves the currently cached tree from Treemon, populating as needed by communicating directly with the device. The tree is locked for the duration of this process, and this prevents Designer from opening the tree. If the value is: • `none` – No configuration information is currently required. This is typical in a steady-state condition. • `cache` – Only the currently cached configuration is required. This is typically seen at startup. • A number – The Log Inserter needs specific information and that number of ION objects has been explicitly requested.

Column	Description

RequestStatus	The status of the tree requests can include one of the following values:
	
	• `ready` – The Log Inserter does not require any configuration information.
	
	• `requesting` – The Log Inserter requires configuration information and is in the process of gathering it. The value in **Request Update Time** indicates how long it has been processing this request.
	
	• `retrying` – A previous tree request was not successful. (See the **Comments** column for the reason.) The request is retried, as shown by the value in **Request Update Time**. The amount of wait time before retrying a request depends on the nature of the unsuccessful tree request:
	    – Tree in use by another node – 10 seconds.
	    – Tree dirty – 10 seconds.
	    – Not responding – 60 seconds.
	    – Tree request timed out after 10 minutes – 5 minutes.
	    – Comm error – 10 seconds.
	    – Other errors – 5 minutes.
	
	• `blocked` – The Log Inserter requires configuration information but all available resources are in use. By default, the Log Inserter can simultaneously request only up to 2 trees per site and 6 trees in total. The **Request Update Time** value indicates how long the request has been pending.
	
	• `processing` – The Log Inserter has received the requested ION objects and is processing them. The **Request Update Time** value indicates how long this request has been processed, including the time during the "requesting" state.
	
	• `abandoned` – This is the same as the `retrying` status but the request of some of the configuration information was not successful following the successful receipt of some information. The Log Inserter recovers when it retries the request.

Request Update Time (s)	The time varies depending on the status of the tree requests described for **RequestStatus**.
pID	The program ID of the program used to poll the current position counters. The Log Inserter now performs its own polling, and as a result, the entry in this column is not used for diagnostic purposes.

The following table summarizes the columns on the **Node Performance** tab:

Column	Description
Node	The name of the device, VIP, or Log Inserter.
Responding	Indicates whether or not the node is responding. For a VIP, this includes all external nodes connected, directly or indirectly, to the input of a Recorder. The responding state is used to determine whether or not the download of the log is caught up.
All Logs Polling Disabled	Indicates if log upload is disabled for all recorders on the device. A `Yes` in the column indicates that log upload is disabled.
TotalLogs	The total number of Data Recorders, Waveform Recorders, Event Log Controllers, and System Log Controllers that the Log Inserter is configured to collect data from a given node. Note that when automatically detecting these modules, this number may change as the Log Inserter gathers configuration information.
PendingRecords	The total number of records that the Log Inserter has requested from the node but has not yet received.
OutstandingRecords	The total number of records not yet uploaded based on the last read position counter on the device and the position of the last uploaded record, taking into account the maximum depth of each log.
ProcessedRecords	The number of records that have been inserted into the database. Note that a record typically corresponds to a number of DataLog entries. The term "record" refers to records at the device level.
Generated Rec. per sec	An estimate of the number of new records being generated per second.
Retrieved Rec. per sec	An estimate of the number of records being uploaded per second.
Avg Retrieval Time (s)	The average round-trip time in seconds taken to retrieve a record from a device.
Avg Processing Time (s)	The average time in seconds necessary to insert a record into the database.
RestoredLogs	The total number of logs that the Log Inserter is configured to gather information for.
ManagedLogs	The total number from the value in **RestoredLogs** that is being monitored by an enabled Log Acquisition Module (LAM).
ConfiguredLogs	The total number from the value in **RestoredLogs** that are Recorders and have source inputs or are Event Log Controllers or System Log Controllers.
ConfirmedLogs	The total number from the value in **RestoredLogs** for which the current configuration is known.
NumCaughtUp	The total number from the value in **RestoredLogs** for which the node is responding and there are no records outstanding or pending.

The following table summarizes the columns on the **Log Performance** tab:

Column	Description
Node	The name of the device, VIP, or Log Inserter in question.
LogHandle	The handle of the Log Register or Event Log Register for this Node.
Responding	Indicates whether or not the node is responding. For a VIP, this includes all external nodes connected, directly or indirectly, to the input of a Recorder. This state is used to determine whether or not it is caught up.
Polling Disabled	Indicates which individual recorders are excluded from polling requests. A `Yes` in the column indicates which recorders are excluded.
PendingRecords	The total number of records that the Log Inserter has requested from the node but has not yet received. This number includes event records that have been uploaded but are cached internally pending configuration information necessary to complete the processing of the cause and/or effect ION objects.
OutstandingRecords	The total number of records not yet uploaded based on the last read position counter on the device and the position of the last uploaded record, taking into account the maximum depth of each log.
ProcessedRecords	The number of records that have been inserted into the database. Note that a record typically corresponds to a number of DataLog entries. In this context, "record" refers to records at the device level.
Generated Rec. per sec	An estimate of the number of new records being generated per second.
Retrieved Rec. per sec	An estimate of the number of records being uploaded per second.
Avg Retrieval Time (s)	The average round-trip time in seconds taken to retrieve a record from a device.
Avg Processing Time (s)	The average time in seconds necessary to insert a record into the database.
Restored	This is always yes. If the log is not "restored", it does not appear in the list.
Managed	A Log Acquisition Module (LAM) is enabled that is monitoring this log.
Configured	The log is a Recorder that has source inputs or it is an Event Log Controller or a System Log Controller.
Confirmed	The latest configuration for the log has been uploaded. For a VIP Recorder that references external devices, directly or indirectly, the configuration information includes information from the external device.
CaughtUp	The node is communicating, the current configuration is known, and there are no outstanding or pending records. For a VIP, any device on which the log depends for information must also be responding.

Alarm Service

Alarm Service provides the status of alarms that you configure and enable in the Software Alarms application.

The information is organized in a grid. The column labels indicate the type of information provided, such as **Rule Name**, **Alarm Name**, **Alarm Status**, and so on. See the *Software Alarms Help* (accessible from the Software Alarms application) for further information about configuring alarms for multiple sources and measurements.

Log Pipeline Service

The Log Pipeline Service diagnostics provides information on the Log Subsystem Data Pipeline. It shows a variety of statistics on log collection and insertion performance.

Log Inserter writes log data into a message queue instead of writing it to SQL Server directly. Another process (the Log Subsystem Router Service) reads the messages from the queue and writes the data to SQL Server.

Previously, the Log Inserter would wait for database writes to complete before processing the next piece of data. This effectively limited the rate of data insertion to something that SQL Server could handle. Writing to MSMQ is much faster: MSMQ can store messages in the queue faster than Log Inserter can retrieve them from the devices. However the performance of SQL Server has not changed, which means that data can accumulate in the queue faster than it can be inserted into the database. Allowing the message queues to become full (they have a limited storage capacity) results in failure modes that are difficult to handle automatically. To avoid this scenario we monitor the size of the inbound data message queue and prevent writes when it contains more than a set number of bytes. If the message queue reaches a specified capacity, then no future messages will be accepted from LogInserter and the write thread is "put on hold" until the message queue has dropped below a specified capacity. This ensures that LogInserter never considers data written that may be missed in the message queue due to over capacity.

The following table summarizes the columns on the **Pipeline Status** tab:

Column	Description
Name	Identifies the message queue (Primary or Secondary) and the type of data being tracked.
Duration	Total time that the diagnostics have been counted. In practice this value will be identical for all rows.
Message Count	Number of messages that have been processed from this queue since the service was started.
Messages Per Minute	Number of messages that have been processed in the last minute.
Messages Per Minute Average	Average of the messages per minute over the last hour (60 samples).
Messages Per Minute Max	Maximum number of messages per minute over the last hour (60 samples).
Messages Per Second	Number of messages that have been processed in the last second.

Column	Description
Messages Per Second Average	Average of the messages per second over the last minute (60 samples).
Messages Per Second Max	Maximum number of messages per second over the last minute (60 samples).
Processing Time Milliseconds Average	Average time taken to process each message (milliseconds).
Processing Time Milliseconds Max	Maximum message processing time (milliseconds).
Time Since Last	Elapsed time since a message was last processed.
Start Time Utc	Time in UTC at which the service was started.

The following table summarizes the columns on the **Message Queues** tab:

Column	Description
Identity	Identity of the message queue.
Queue Type	Data or Control
Message Count	Number of messages currently in the queue.
Message Kilobytes	Size of messages currently in the queue (kilobytes).
Maximum Kilobytes	Maximum size allowed for the queue. Note that a value of 4294967295.00 (4GB) is the maximum amount of data that the entire MSMQ service can support. This effectively indicates that no queue-specific limit has been defined.

The **Message Queues** tab allows you to inspect messages flowing through the pipeline. Enabling message capture increases the load on the pipeline so this should not be left activated indefinitely.

The controls on the top select which messages are written to the table. This filtering only occurs as the messages arrive at the tab; once a message is displayed in the table it will not be removed when the Message Type or Source Filtering fields are changed. Rows already in the table may be filtered by the fields below the header.

The viewer buffers messages as they arrive to avoid locking up the receiving thread. If the buffer fills (which indicates that messages are arriving faster than the viewer can process them) a message indicating how many messages were missed will be written to the table.

Communications Diagnostics

Communications Diagnostics provides diagnostics information for sites and devices connected to the workstation.

Site overview

Diagnostics information for the sites are contained in these tabs:

- **Device Summary** displays communications statistics for each site.

- **NetUser Status** displays the number of ION programs currently in the ION Network Router Service queue (awaiting processing) and the total number of ION programs already processed.

> **NOTE**: Requests and responses transmitted between the software components are referred to as "ION programs".

Site/Device Diagnostics

Diagnostics information for sites and devices are summarized in these tabs:

- **Communication Status** displays error rates and connection statistics for the selected site or device. The following information is available from the **Communications Status** tab:

Column	Description
Node	The device (or software node) name.
Requests	The number of communications requests transmitted to the meter.
Responses	The number of successful responses received.
Request Ratio	The number of requests sent to the device to fulfil the last client request. The value is always 1 for ION devices but it varies for Modbus devices.
Total Errors	The total number of communication errors.
Total Err Rate (%)	The ratio of Total Errors to Requests.
Sliding Err Rate (%)	The error rate in the last 100 requests. This can indicate a trend in communications performance.
Time Util (%)	The percentage of the communication channel utilized (serial line or Ethernet) on the site.
Avg Resp Time (s)	Average time in seconds for the meter to respond.
Last Resp Time (s)	The last response time, in seconds.
Timeouts	The number of timeouts. A timeout occurs when no data is received in response to a request.
Bad CRC	The number of bad packets received, that is, those that do not pass the error-detection checksum.
Incompl. Frm	The number of incomplete packets received, that is, those that did not have all the expected bytes.
Broken Conn.	Number of times the connection was lost to the meters on a site.
Bad Frames	The number of received packets that had an internal error.

Column	Description
HW Errors	Number of errors reported by the computer's communication hardware.
Misc Errors	Number of other errors that do not fit any of the above descriptions.

- **Site Status** displays site statistics such as connection status and totals.

- **Polling Status** displays the number of programs currently in the WinPM.Net Site Service queue (awaiting processing) and the total number of programs already processed.

Communication Status vs. Site Status

This section explains the difference between the statistics provided on the **Communication Status** tab and those on the **Site Status** tab.

"Total Errors" in the **Communication Status** tab is an WinPM.Net Site Service derived statistic, while "Bad Responses" in the **Site Status** tab is a client derived statistic.

To explain this difference, consider a situation where a direct site is experiencing timeouts. Communications with the device is attempted according to two parameters: **Connect Attempts** (an advanced site property) and **Maximum Attempts Multiple** (an advanced device property). Multiplying the values of these two properties determines the number of attempts made to re-establish communications with the device.

For instance, if **Connect Attempts** is set to 1 and **Maximum Attempts Multiple** is set to 3, the device will go offline after 3 attempts (that is, 1 x 3).

The "Total Errors" statistic increases by one every time WinPM.Net Site Service detects a timeout. However, the "Bad Responses" statistic only increments every time a response is sent back to a client.

Using the previous example, consider the case where four timeouts occurred and the device went offline. In this case, "Total Errors" increases by four, while "Bad Responses" only increases by one. If only two timeouts occurred, "Total Errors" would increase by two, while "Bad Responses" would not change.

The following information is available from the **Site Status** tab.

Column	Description
Node	The device (or software node) name.
Status	The device communication status.
Current Attempt	The current number of repeated attempts to communicate with the device.
Max Attempts	The maximum number of attempts before flagging the device as offline (Timed-out).
Offline Count	The total number of times the device went offline.
Bad Responses	The total number of errors sent back to the clients, such as to Vista.

Column	Description
Last Response	The time when the last response was received.
Last Attempt	The last time that a request was sent to the device.
RT Data Reqs	The total number of requests to the device sent by the Real Time Data Service.
TreeMon Reqs	The total number of requests to the device sent by the TreeMon service.
VISTA Reqs	The total number of one shot requests to the device sent by a Vista client (control, label requests...).
LogInserter Reqs	The total number of requests to the device sent by the LogInserter service.
IONSERVICE Reqs	The total number of requests to the device sent by ION real-time services.

Note that the last five columns on the **Site Status** tab are dynamic. That is, the columns are only shown when requests were sent to the device from a WinPM.Net service or client.

Additional commands

The following sections describe additional display options and shortcut menus available in Diagnostics Viewer.

Diagnostic Details

In the tabs on the diagnostics information pane, double-click a row to display its **Diagnostic Details** screen. This displays the diagnostic information for the selected item only.

Use the **Previous** and **Next** buttons to view the details of other rows in that tab of the diagnostics information pane.

To copy information to the clipboard, select the rows you want to copy, then press **CTRL+C**.

Diagnostics Information pane shortcut menu options

Right-click the diagnostics information pane to display a shortcut menu. The following table lists all the commands available (though not all panes in Diagnostics Viewer provide all the commands listed):

Right-click Option	Description
Update	Refreshes the information in the diagnostic table.
Reset	Resets the information in the diagnostic table (not available in the Communications Server Diagnostics display).
Copy All	Copies all selected information to the clipboard.

Right-click Option	Description
Auto Scroll	Enabled by default, this option is only available in the Console Messages tab of the Communications Server Diagnostics display. This option automatically scrolls and selects the latest console message. Clear this option to disable scrolling (that is., select and view an older console message without jumping to the latest one when Diagnostics Viewer refreshes).
Options	Displays the **Options** dialog where you can change the diagnostics refresh rate. Note that changing the refresh rate frequency can affect the product's performance.

Event Watcher Manager

The Event Watcher Manager defines events for the "On Event" selection in report subscriptions. (For more information about report subscriptions, see *Creating report subscriptions* in the online help for Reports in the Web Applications component.)

The Event Watcher Manager interface consists of two tabs:

- **Event Watchers**: An event watcher links a device or group of devices to the conditions defined in a template.

- **Templates**: A template defines the conditions that make up an event.

TIP: For instructions on using filtering, sorting, and column selection to customize the Event Watcher Manager interface, see [Customizing and navigating interface displays](#).

Opening Event Watcher Manager

Open Management Console and select **Tools > Event Watcher**.

Symbols used in the Updated column

The following symbols are used in the **Updated** column to display the status of information in a row:

*	Information in the row has been modified since it was last saved.
!	Information in the row is inaccurate or incomplete.
+	The row is new since the user or group information was last saved.

Creating a template

Before adding an event watcher, you must have a template that defines the conditions to be watched. You can use one of the default templates or create your own. Use the **Templates** tab to define and edit templates.

Event Watcher Manager comes with default templates that can be modified for specific devices. Test any changes to the default template to ensure that events are available for selection for report subscriptions. In cases where a device is not supported by the default templates, you can create custom templates to define what you need.

To define a custom template:

1. Click **New** in the top right corner of the tab. A new row appears in the main table for the new template.

2. Type a name for the template. Type a description for the template if desired.

3. Use the Template Conditions section to set the conditions to be watched. See Using the Template Conditions section, below.

4. Click **Apply** to save your changes, click **OK** to save your changes and exit Event Watcher Manager, or click the **Event Watchers** tab to set up an event watcher.

Using the Template Conditions section

Use this section of the interface to configure the condition the template watches for.

1. Add a condition:

   - To add a condition from the event log, click **Select from Event Log** to open the **Select Event Log Lines** dialog. To select an event, click on the boxes at the left of the table to select a check box. Click on the check box again to clear the check mark. Click **OK** to return to the Templates tab.

   - To add a condition without selecting from the event log, click **New**. A new row appears in the table for the new condition.

   NOTE: It is recommended that you use **Select From Event Log** to add conditions. This helps ensure that the syntax is correct. If the syntax is not correct, the event watcher will not work. Conditions only appear in the Select Event Log Lines dialog after they have occurred in the system.

2. Select the parameters you want to include in the condition by selecting or clearing the check boxes below the Template Conditions table. The available parameters are: Priority, Cause, Cause Value, Effect, Effect Value. You must include at least one parameter. If you select multiple parameters, the event watcher only activates when all parameters are met.

The Logic of the Template Conditions

The conditions within a row of the Template Conditions table must all be true for the condition to be met. For example, if you have a row with a Priority of greater than 220 and an Effect Value of Disturbance, both of these must be met for the condition to be true. If you have multiple rows in the table, the condition is met if any of the rows are true. For example, if you have a row with Priority greater than 220 and a row with an Effect Value of Disturbance, the condition is met if either of these are true.

Deleting and reverting

To delete a template condition, select the condition from the Template Condition table and click the **Delete** button in the Edit <Template> section. To delete a template, select the template from the main table and click the **Delete** button in the top right corner of the tab.

To undo changes, click **Revert**.

Adding an event watcher

To add an event watcher, you need a template. You can use one of the default templates or define your own (see Creating a template). Use the **Event Watchers** tab to define and edit event watchers.

1. Click **New** to add a new event watcher. A new row appears in the main table for the new event. Use the Event Watcher Editing Area to define the event watcher.

2. Type a name for the event watcher.

3. Select the template you want to use from the Template dropdown list.

4. Type a description of the event watcher if desired.

5. From the Devices table, select the devices you want to include in the event watcher. Click the check boxes to the left side of the devices to select or clear a device.

   > **NOTE**: Checking **Select All** will include any Query Server, Log Inserter or Virtual Processor instances in the system. It also includes any devices or software nodes added in the future.

6. Click **Apply** to save your changes, click **OK** to save your changes and exit Event Watcher Manager, or click the **Templates** tab to edit the templates.

Deleting and reverting

To delete an event watcher, select the event watcher from the table and click **Delete**.

To undo changes, click **Revert**.

Hierarchy Manager

Depending on the number of devices you are currently monitoring, organizing the data coming from those devices can be a difficult task. The Hierarchy Manager feature allows you to organize the devices in WinPM.Net by defining their relationships and displaying them in an easy to understand way. Once these relationships have been created, energy data can be grouped and summarized, and used by other components of WinPM.Net.

Starting Hierarchy Manager

To use Hierarchy Manager:

1. Open Management Console.

2. Enter an operator-level or supervisor-level access username and password and click **OK**.

3. Click **Tools > Web Tools > Hierarchy Manager** to open the Hierarchy Manager Login dialog.

4. Enter an operator-level or supervisor-level access username and password and click **OK**.

   Hierarchy Manager opens using the default hierarchy template.

The hierarchy creation process is dependent on the current template. See the Hierarchy Manager online help for information regarding this feature. Click the **Help** link in the top right corner of Hierarchy Manager to access the information.

License Manager

For additional device or client user support (for example, for Vista clients) in your system, or to add OPC options, order the appropriate license from Siemens Industry.

When you receive your new license, use License Manager to upgrade the number of supported devices and client users in your system:

1. Log in to Management Console.

2. Click **Tools > License Manager** to open the **License Manager** dialog.

    The current product key number, number of devices in the system, and maximum number of devices supported are listed in the System Info section.

3. Type the new product key in the Product Key field.

4. Click **OK**.

The System Info section lists the updated support information from the newly activated product key. If the new key supports additional features or options, these are made available immediately, without restarting Management Console.

Logical Device editors

The logical device functionality allows you to assign logical names to output registers on physical devices, and associate the generic inputs with standard measurements, including WAGES measurements.

Before you can create a logical device, the physical device that it references must be created. You can create one or more logical devices from a single physical device, or you can also create a single logical device from several physical devices.

After you create a logical device, you can view its real-time data in the following components:

*   Vista.

*   The Diagrams application in the Web Applications component.

⚠ CAUTION

INACCURATE DATA RESULTS

*   Do not incorrectly configure the software or the devices.

*   Do not base your maintenance or service actions solely on messages and information displayed by the software.

*   Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.

Failure to follow these instructions can result in injury or equipment damage.

Each logical device has a logical device type associated with it. Each logical device type is defined with a collection of output measurements. The measurements defined by the device type become the output measurements for instances of that device type, that is, for the logical devices.

You use the Logical Device Type Editor to create logical device types. See <u>Using the Logical Device Type Editor</u> for information about viewing, creating, editing, and deleting logical device types.

The Logical Device Editor in Management Console allows you to configure specific logical devices by mapping input registers to output measurements that are defined in the associated device type. See <u>Using the Logical Device Editor</u> for information about creating, configuring, editing, and deleting logical devices.

Open the Logical Device editor applications from the **Tools** menu in the Management Console.

> **NOTE**: Access to this application or function is controlled by user privileges. See **Default User Access Level Privileges** in Web Applications help, and **Customizing Access Level Privileges** in User Manager help for details.

Using the Logical Device Type Editor

Use the Logical Device Type Editor to define a collection of output measurements for each logical device type. Logical device types created with this editor are available for selection in the Logical Device Editor, where you map input registers to output measurements.

Open the Logical Device Type Editor

1. Start Management Console and log in using an account with supervisor-level access.

2. Select **Tools > Logical Device > Logical Device Type Editor**.

The top portion of the editor contains a logical device type display grid listing the logical device types that have been defined. The bottom portion is an edit area where you can modify existing device types and their measurements, or create new ones.
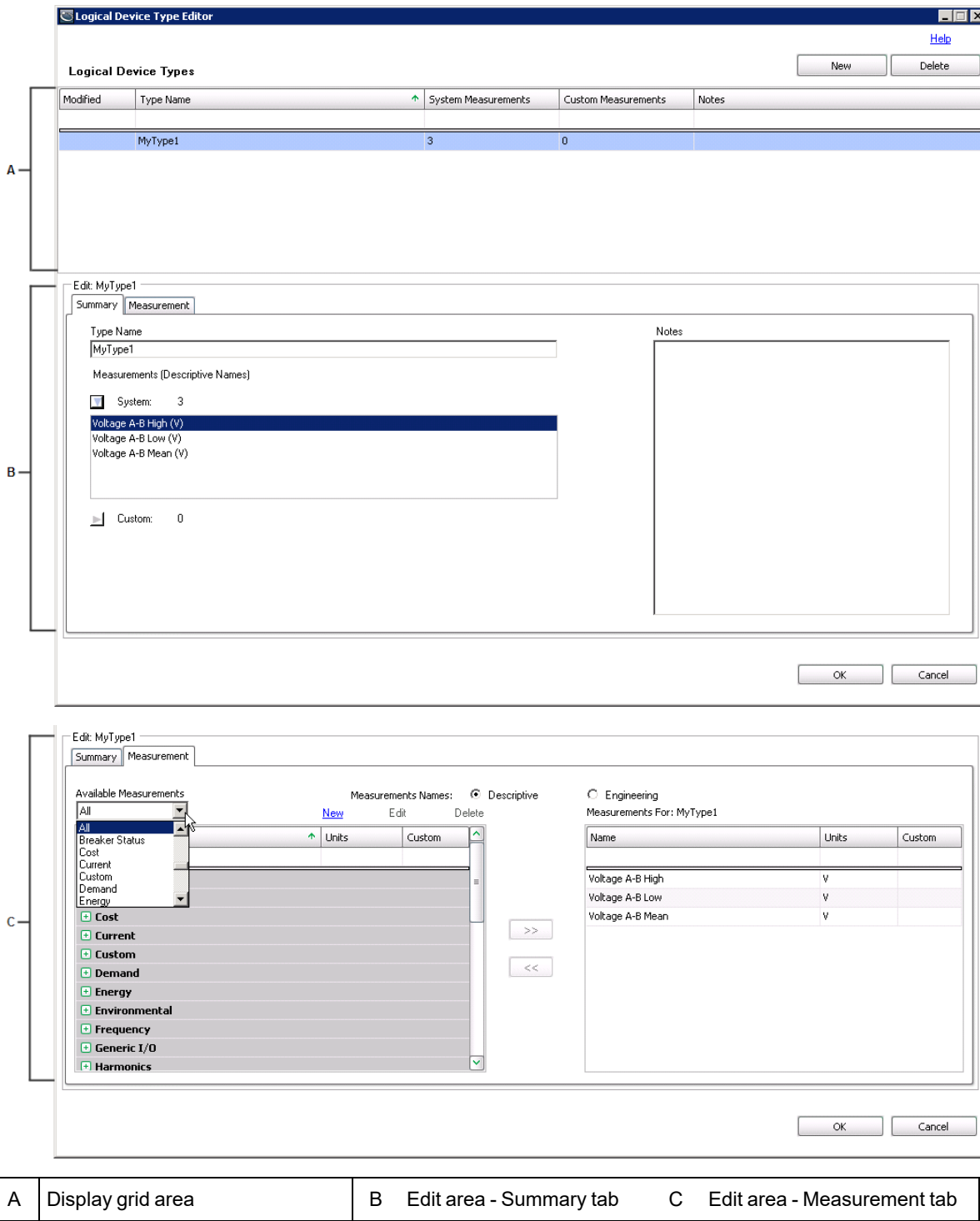
| A | Display grid area | B | Edit area - Summary tab | C | Edit area - Measurement tab |

The display grid area

The **Modified** column displays one of the following characters indicating a change of status of an entry:

+ (plus) - a new entry.

* (asterisk) - an existing entry has been modified.

! (exclamation mark) - the entry is incomplete or in error.

The **Type Name** column contains the names for the defined device types. Type names must be unique.

The **System Measurements** and **Custom Measurements** columns show the number of measurements that have been defined for each device type.

The edit area

When a device type is selected in the display grid, the current information for that device type is displayed in the edit area. The edit area has a **Summary** tab and a **Measurement** tab.

The **Summary** tab contains the following:

- A **Type Name** field showing the name of the device type that is selected in the display grid at the top.

- In the **Measurements** area, expand **System** and **Custom** to view the measurements defined for these respective categories. You can expand each list to show all of the measurements that have been defined for the selected device type.

- A **Notes** input field that you can use to describe the device type.

The **Measurement** tab contains the following:

- Options that indicate whether to display the measurements with descriptive names or engineering names.

- An **Available Measurements** dropdown list that contains category names for the measurements, such as **Air**, **Current**, and **Voltage**. When you select one of these categories, only those measurements associated with the selected category appear in the grid.

  The default **All** in the list displays all of the categories and their respective measurements in the grid.

- An available measurements grid listing the measurements available for selection when you define a device type. Measurements that are grayed out indicate that they have been added to the selected device type. Clicking **New** opens a dialog for defining a new, custom measurement. An X in the **Custom** column identifies the entry as a custom measurement. The **Edit** and **Delete** links give you the option to modify or delete custom measurements only.

  NOTE: It is recommended that you add a new custom measurement *only if* an existing measurement does not suffice. Factory-defined System Measurements contain considerable information about their type and usage, which is then used throughout the system to provide intelligent assistance in Dashboard controls and Report selections. Since custom measurements do not have this information, their usage may not provide the expected results.

- A **Measurements For** grid that lists the measurements that comprise the device type that is selected in the display grid area. Use the **>>** button to add a measurement that you selected in the list of available measurements to the **Measurements For** list for the device type. Measurements that you select for a device type are grayed out to indicate that they have been assigned to the device type. Use the **<<** button to remove a selected measurement from the **Measurements For** grid. Measurements that are removed are no longer grayed out in the list of available measurements and they are available for selection.

Note that if you attempt to delete a custom measurement that has been mapped to any logical device, a message indicates that the measurement cannot be deleted. You need to unmap the custom measurement from the logical devices before you can delete it.

In all grids:

- Each of the columns can be sorted in ascending or descending order when you click on the column heading.

- When you place your cursor on a column heading, a filter icon appears on the right side of the heading area. Click this icon to open a list of items that are available for filtering. Click the item that you want to use as the filter. When you do this, the filter condition displays on the left side of the status area at the bottom of the grid. Click the X icon in the status area to turn off the filter.

- The blank area immediately below a column heading is an entry field for filtering on an entry you type in that area. For example, to filter on a specific name, enter a name under the **Name** column to automatically filter the entries in the grid.

- You can drag and drop the grid column headers to rearrange the order of the columns in the grid.

Creating a new logical device type

Complete the following steps to create a new logical device type:

1. Click **New** on the top right of the editor.

   This clears the **Type Name** field and resets **System measurements** and **Custom measurements** to 0 (zero) on the **Summary** tab.

   All is selected by default in the **Available Measurements** list on the **Measurements** tab.

2. Enter a device type name in the **Type Name** field.

   The name appears in the logical device type display grid and an ! (exclamation) appears in the **Modified** column to indicate that the device type entry is incomplete.

3. Click the **Measurements** tab to go to the list of measurements that are available for selection.

4. Locate and click an available measurement that you want to associate with the device type, then click **>>** to add it to the **Measurements For** list. Repeat this step for each measurement that you want to associate with the device type.

5. To delete a device type, click the device type in the display grid and click **Delete** on the top right of the editor.

   Note that **Delete** is disabled if any logical devices are associated with the logical device type.

Custom measurements

The following sections describe how to create, modify, or delete custom measurements. Because factory-defined System Measurements contain considerable information about their type and usage, custom measurements should only be created in exceptional circumstances, and only with an understanding of the expected results from their usage.

⚠ CAUTION

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.

- Do not base your maintenance or service actions solely on messages and information displayed by the software.

- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.

Failure to follow these instructions can result in injury or equipment damage.

Creating a new custom measurement

Complete the following steps to create a new, custom measurement:

1. Click **New** beside the **Available Measurements** dropdown list to open the **New Measurement** dialog. (You can also edit a new custom measurement by clicking **Edit** and modifying the entries.)

2. Enter a descriptive name and an engineering name for the new measurement in the respective fields. For example, `Voltage A-B Low` is the descriptive name, and `Vll ab low` is the engineering name.

3. Select the unit of measure from the **Units** list or add a custom unit of measure by typing it in the **Units** field.

   Note that the combination of the descriptive name, the engineering name, and the unit of measure must be unique for each custom measurement.

4. Select a type of information to be measured from the **Quantity** list.

5. Click **OK** to add the measurement.

   The new measurement is added as an available measurement in the grid and an X in the **Custom** column indicates that it is a custom measurement.

   You need to select **All** or **Custom** in the **Available Measurements** list to see the custom measurement in the grid.

Modifying or deleting a custom measurement

Complete the following steps to modify a custom measurement. Note that you cannot modify or delete system measurements.

1. Go to the **Measurements** tab of the Logical Device Type Editor.

2. Select the custom measurement that you want to modify or delete.

   Custom measurements that have been associated with a logical device type can be edited but not deleted. Selecting **Delete** results in a message indicating that the custom measurement cannot be deleted because it is mapped either to a logical device type or to a logical device that is logging data.

3. To modify the custom measurement, click **Edit** to open the **Edit Measurement** dialog, update the entries or selections in the dialog, and click **OK** to apply to changes.

4. To delete a custom measurement, click **Delete**.

    A message confirms the deletion.

    Note that if you attempt to delete a custom measurement that is associated with any logical devices, a message indicates that the measurement cannot be deleted.

Using the Logical Device Editor

Use the Logical Device Editor to map measurements from physical devices to measurements provided by the logical device type that is associated with a logical device.

To open the Logical Device Editor:

1. Start Management Console and log in using an account with supervisor-level access.

2. Click the **Devices** icon, then right-click the Display window and select **New > Logical Device**.

    The devices defined in Management Console are listed in the **Input Devices** area of the Logical Device Editor.

The top portion of the editor consists of a header area that contains instructions, selection lists, and input fields. The bottom portion is the mapping area that allows you to map the input registers of a physical device to a list of measurements provided by the associated device type.
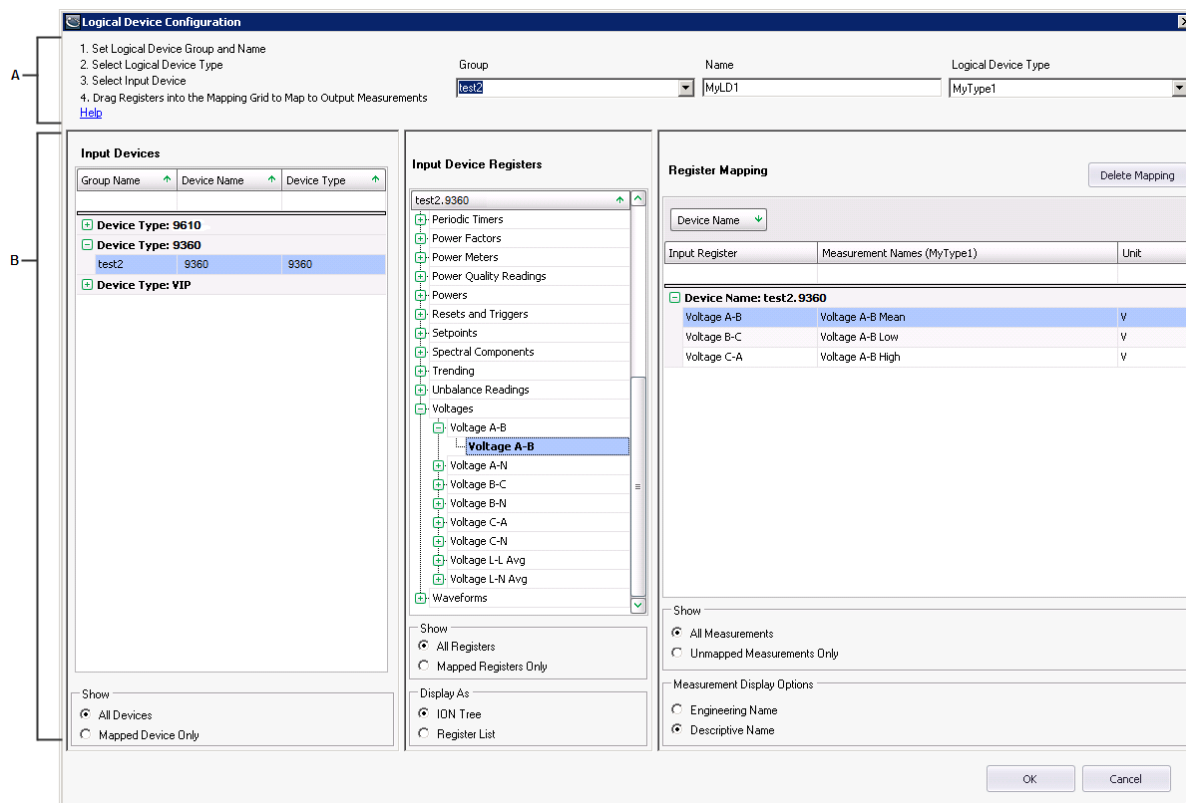
A	Header area	B	Mapping area

The header area

The instructions indicate the steps required to create a logical device.

The **Group** field is a combination input field and dropdown list. You can either enter a name for the group or select a group from the list.

The **Name** field identifies the name of the logical device that you are defining. Enter the name of the logical device in the field.

The **Logical Device Type** dropdown list contains the names of logical device types that are available for selection.

The mapping area

The **Input Devices** grid lists the devices that are in your system. When you select a device, its output registers are listed in the **Input Device Registers** area.

The **Register Mapping** grid is populated with a list of the output measurements that are defined for the logical device type that you select in the **Logical Device Type** dropdown list. They are listed under a generic **Device Name** label pending their mapping to the logical device name that you specified. Drag an entry from the **Input Device Registers** list to an output measurement in the **Register Mapping** grid to map the two values.

Defining a logical device

1. Select a logical device type from the **Logical Device Type** dropdown list.

    The output measurements defined for the selected logical device type are listed in the **Output Measurement** column in the **Register Mapping** area.

2. Either select an existing group name from the **Group** dropdown list, or enter a group name in the field.

3. Enter a name for the new logical device in the **Name** field.

4. Select one of the input devices from the **Input Devices** area of the editor.

    The registers for the selected input device appear in the **Input Device Registers** area. Note that if you choose to display the registers for the input device in an ION tree format, delays in displaying the registers can occur for devices with a large volume of registers.

5. Map an input device register to an output measurement by dragging it to the output measurement listed in the **Register Mapping** area of the editor.

    The input device registers appear in bold font when they are mapped to an output measurement.

6. Click **OK** to save the logical device configuration in Management Console.

    Any unmapped measurements are displayed in red.

7. To edit a logical device that is in the **Device** panel of Management Console, double-click the device name, or right-click the device name and select **Configure Device** to open the Logical Device Editor, or click the device then select **Edit > Configure Device**.

    Note that:

    -   If you choose to edit the configuration of the logical device, delays can occur in displaying the registers for devices with a large volume of configured registers.

- **Duplicate and Configure** in Management Console's **Edit** menu cannot be used with logical devices. See Creating multiple logical devices for information about creating more than one logical device.

Considerations regarding logical devices and historical data

When a logical device is configured, historical data for an output register (that is, an output measurement) on the physical device is automatically routed to the corresponding output measurement on the logical device.

When a logical device is configured, historical data is logged to the logical device and not the related physical device.

If multiple logical devices include the same input measurement from the same physical device, historical data from that input measurement is routed to the logical device that was configured last.

Deleting a physical device that is mapped to a logical device

If a physical device that is mapped to a logical device is deleted, the mapping of input measurements for the logical device can no longer be resolved. A message dialog opens to indicate the result of the delete action and asks whether you want to continue with the deletion.

If you right-click the logical device in Management Console and select **Configure Device** to open the Logical Device Editor after you have deleted the physical device, a message indicates that the physical device has been deleted. The **Register Mapping** grid for the logical device shows the deleted physical device/ION handle in the **Input Register** column.

If you re-create the physical device, the input measurement mapping for the logical device is automatically restored.

Renaming a physical device that is mapped to a logical device

If a physical device that is mapped to a logical device is renamed, a message dialog opens to indicate the result of the rename action and asks whether you want to continue with renaming.

If you right-click the logical device in Management Console and select **Configure Device** to open the Logical Device Editor after you have renamed the physical device, the **Register Mapping** grid for the logical device shows the renamed physical device in the **Device Name** column.

Deleting a site which has a physical device mapped to a logical device

If a site with a physical device mapped to a logical device is deleted, the physical device is also deleted and the mapping of input measurements for the logical device can no longer be resolved.

If you right-click the logical device in Management Console and select **Configure Device** to open the Logical Device Editor after you have deleted the site, a message indicates that the physical device has been deleted. The **Register Mapping** grid for the logical device shows the deleted physical device/ION handle in the **Input Register** column.

Creating multiple logical devices

You can create multiple logical devices by:

- Exporting a logical device configuration to a comma-separated values (CSV) file.

- Modifying the CSV file to add multiple logical device configurations based on the original

logical device configuration.

- Importing the CSV file in Management Console to add the logical devices to the system.

Export a logical device configuration

Complete the following steps to export one or more logical device configurations to a CSV file.

1. Use the Logical Device Editor to create at least one logical device in the Devices area of Management Console.

2. Select one or more logical devices, right-click and select **Export** to open the **Export Status** dialog.

    Note that if you choose to export the registers for the logical device, delays in the export process can occur for logical devices with a large volume of configured registers.

3. Click the browse button on the right of the **Selected File** field to open the **Save As** dialog.

4. Navigate to the location where you want to save the file and enter a name for the file in the **File name** field.

    `Text files (*.csv)` is pre-selected in the **Save as type** field.

5. Click **Save** to return to the **Export Status** dialog.

    The **Selected File** field is populated with the location and file name that you specified.

6. Click **Export**.

    The **Status** column in the grid area of the dialog indicates **Exported** for a completed export operation. If the export operation is not successful, for example if the input registers cannot be retrieved for the logical device being exported, it is indicated in the **Status** column. Click the link to open the log file to view additional information.

    Configuration information for all of the selected logical devices is exported to the CSV file.

Create multiple entries in the CSV file

Use Microsoft Excel or a similar spreadsheet editor to open the CSV file. The column headings in the spreadsheet are labeled as follows:

- Logical device name

- Logical device type

- Physical device name

- Physical device type

- Input register

- Output measurement

- Handle - this column heading is required but the entries are not required unless there are duplicate labels, that is, input registers with the same name but with a different ION handle. In this case the ION handle is required. You can view the ION handles by exporting the input measurements from the Logical Device Type Editor.

NOTE: The column headings remain in English for localized versions of the product. This is required for CSV file-processing purposes.

To create multiple entries for the logical device configuration, use the copy and paste function in Excel. It is recommended that only users with supervisor-level access create multiple entries.

1. Select the rows that you want to duplicate and press Ctrl+C to copy them.

2. Place the cursor in the left-most cell of the first blank row and press Ctrl+V to paste copies of the selected rows.

3. Continue the paste operation for as many logical device configurations that you want to create.

4. After completing the paste operation, be sure that you use unique device names to avoid duplicates.

You can also export all of the register names of a physical device as well as all of the measurement names of a logical device type. This gives you the ability to use these additional CSV files to copy and paste additional register names and measurement names into the CSV file you are using to create the multiple logical devices.

To export all of the register names of a physical device:

1. Open the Logical Device Editor and select a physical device.

2. Right-click anywhere in the register list in the **Input Device Registers** area to open a menu and select **Export Device Input Registers**.

3. Enter a name for the file and click **OK**.

To export all of the measurement names of a logical device type:

1. Open Management Console and select **Tools > System >** Logical Device Type Editor.

2. Select a logical device type in the logical device types area of the editor.

3. Click the **Measurements** tab.

4. Right-click anywhere in the grid under **Measurements For** and select **Export Measurements**.

5. Enter a name for the file and click **OK**.

Import multiple logical device configurations

Complete the following steps to import the CSV file with the multiple logical device configurations. It is recommended that only users with supervisor-level access import logical device configurations.

1. Open Management Console and select **Tools > Import Logical Device** in the **Devices** panel of Management Console to open the **Logical Device Import Form** window.

   Logical Device Import Form is available only when you are in the **Devices** panel of Management Console,

2. Use the Browse button to navigate to and select the CSV file to import.

3. Select **Overwrite Existing Devices** if you want overwrite logical devices with the same name that were previously created.

   Logical devices with the same name can occur as a result of a previous import operation.

4. Click **Import** to add the multiple logical devices to the Devices area of Management Console.

   The system performs a validation of the configuration data during the import operation. If an error occurs during the import operation:

   – The **Status** column in the grid area indicates **Failed** for any logical device that is not imported. Click **Failed** to view details about the cause of the unsuccessful import.

   – Logical devices are created for all correct entries in the CSV file. Logical device entries that are incorrect are not created in Management Console.

Manual Data Editor

Use Manual Data Editor to edit logged measurement data or to manually enter measurement data into the WinPM.Net database. With this tool, you can adjust incorrect data, outliers, roll overs, jitter, and other unexpected or unwanted data records. You can fill data gaps or enter manually collected data, such as clipboard readings from mechanical meters. The Manual Data Editor also has the ability to create new, custom measurements in WinPM.Net.

NOTE: Manual Data Editor can only be used to edit trend data logs. It cannot be used to edit waveform logs or event logs.

NOTICE
LOSS OF DATA
Back up the database before making manual database edits.
Failure to follow these instructions can result in permanent loss of data.

Editing existing logged measurement data

Edit existing data logs to adjust incorrect data, outliers, roll overs, jitter, and other unexpected or unwanted data. You can also fill in missing data and insert missing data records.

To edit existing logged measurement data:

1. In Manual Data Editor, click **Load New Data** to open the Load Data dialog box.

   NOTE: Load Data opens automatically when you start Manual Data Editor.

2. In Load Data:
   – Select the Source for which you want to edit the logged data.
   – Select the Measurement for which you want to edit the logged data.
   – Click **Add Data to Selection** to add the source/measurement to the list of items to be opened in the data editor.

3. (Optional) Repeat step 2 to add additional sources and measurements to the selection list.

4. In Load Data, select the **Time Range** for the logged data you want to open in the data editor.

5. (Optional) In Load Data, set the following data editor options:

   – **Timezone** - Set the timezone for the display of the configurable timestamp in the data editor. The data editor shows two timestamps for each data record, one timestamp in UTC and one in a configurable timezone.

   – **Timestamp Format** - Set the format for the timestamp display in the data editor.

   – **Number Format** - Set the number of decimals that are displayed for the logged data in the data editor.

   – **Is Interval Data** - Select the check box to enable this property. Set the expected logging interval for the data.

   This setting ensures that there is a data row in the editor for every expected log entry. If there is no data record for an expected entry in the database, then the Manual Data Editor inserts a timestamped row with a <no data> data values in the editor. This makes it easier to identify missing records and correct them.

   > **NOTE**: The data editor shows all logged records in the selected time range, regardless of the Interval Data settings. No records are hidden or filtered out.

6. Click **Load Data** to open the source/measurement data in the data editor.

7. (Optional) In the data editor, click **Load New Data** to change which data is loaded, the time range, or the editor options.

8. (Optional) In the data editor, click **View Issue Report** to see which data issues were identified with the loaded data by Manual Data Editor.

9. (Optional) In the data editor, use **Locate Issues** to find and step through the data issues identified by Manual Data Editor. The data editor uses different color highlights to mark data issues.

10. Locate data records that you want to edit and use any of the following tools or methods to edit the data:

   – Click a data record and edit the data value.

   – **Insert Timestamp** - Insert a new data record with a timestamp that you specify. The timestamp must be in the time range of the data that is loaded into the editor. The new record has <no data> data values.

   > **NOTE**: A historical data log timestamp marks the end of the logging interval. For example, a data log with a 15 minute logging interval and timestamp of 17:00 represents data for the time interval of 16:45 - 17:00.

   – **Smart Gap Fill** - Automatically fill missing data values in a selected time range, or the entire time range of the loaded data set. You can choose between different interpolation algorithms to fill the missing data. See Smart Gap Fill UI for more details.

   – Delete a data record or data value. These options are available in the right-click context menu.

   – Set a data value to <null>. This option is available in the right-click context menu.

- Copy one or more data values and paste them into an external editor, such as Microsoft Excel, for bulk editing. Paste the modified data values back into Manual Data Editor. The copy and paste options are available in the right-click context menu.

   NOTE: Only select the data values for copy/paste, not the timestamps.

   TIP: Modified data values are shown in bold font in the Manual Data Editor.

11. (Optional) In the data editor, use **Locate Changes** to find and step through the edited data records.

12. (Optional) Revert edited data values back to their original value. This option is available in the right-click context menu. Use **Undo all Changes** to revert all edits in the loaded data set.

13. Click **Apply Changes** to write the data log changes to the database.

   NOTE: The Apply Changes command permanently writes the data log changes to the database. There is no undo option for this command. See Reverting previous data edits for more information.

14. (Optional) Click **View Log** to see a historical audit log of all the data log changes.

15. Close Manual Data Editor.

For information on how to use the Manual Data Editor , see:

- Editing existing logged measurement data
- Adding manually collected measurement data
- Viewing previous change details
- Reverting previous data edits
- Creating new measurements

For reference information see:

- Manual Data Editor user interface

Adding manually collected measurement data

Manually add data logs to the database that were collected from data sources that do not have a communication link with the software.

To add manually collected measurement data:

1. In Manual Data Editor, click **Load New Data** to open the Load Data dialog box.

   NOTE: Load Data opens automatically when you start Manual Data Editor.

2. In Load Data:
   - Select the Source for which you want to add data.
   - Select the Measurement for which you want to add data. If the measurement does not exist, click **Create new Measurement** to create a new measurement.

     > **NOTE**: Check the list of available measurements to see if one exists for your application before creating a new measurement.

   - Click **Add Data to Selection** to add the source/measurement to the list of items to be opened in the editor.

3. (Optional) Repeat step 2 to add additional sources and measurements to the selection list.

4. In Load Data, select the **Time Range** for the data you want to add in the data editor.

5. (Optional) In Load Data, set the following data editor options:
   - **Timezone** - Set the timezone for the display of the configurable timestamp in the data editor. The data editor shows two timestamps for each data record, one timestamp in UTC and one in a configurable timezone.
   - **Timestamp Format** - Set the format for the timestamp display in the data editor.
   - **Number Format** - Set the number of decimals that are displayed for the logged data in the data editor.
   - **Is Interval Data** - Select the check box to enable this property. Set the expected logging interval for the data. This setting ensures that there is a data row in the editor for every expected log entry.

6. Click **Load Data** to open the source/measurement data entry fields in the data editor.

7. (Optional) In the data editor, click **Load New Data** to change which data is loaded, the time range, or the editor options.

8. In the data editor enter the measurement data using any of the following tools or methods:
   - Click a data record and edit the data value.
   - **Insert Timestamp** - Insert a new data record with a timestamp that you specify. The timestamp must be in the time range of the data that is loaded into the editor. The new record has <no data> data values.

     > **NOTE**: A historical data log timestamp marks the end of the logging interval. For example, a data log with a 15 minute logging interval and timestamp of 17:00 represents data for the time interval of 16:45 - 17:00. That means that the last data log for a day has a timestamp of midnight (or 12 AM) of the following day. For example, the last data log timestamp for January 31 has a timestamp of February 1, 00:00 hrs (or 12 AM).

   - **Smart Gap Fill** - Automatically fill missing data values in a selected time range, or the entire time range of the loaded data set. You can choose between different interpolation algorithms to fill the missing data. See Smart Gap Fill UI for more details.

- Delete a data record or data value. These options are available in the right-click context menu.

- Set a data value to <null>. This option is available in the right-click context menu.

- Copy one or more data values and paste them into an external editor, such as Microsoft Excel, for bulk editing. Paste the modified data values back into Manual Data Editor. The copy and paste options are available in the right-click context menu.

  NOTE: Only select the data values for copy/paste, not the timestamps.

  TIP: Modified data values are shown in bold font in the Manual Data Editor.

9. Click **Apply Changes** to write the new data log entries to the database.

   NOTE: The Apply Changes command permanently writes the data log changes to the database. There is no undo option for this command. See Reverting previous data edits for more information.

10. (Optional) Click **View Log** to see a historical audit log of all the data log changes.

11. Close Manual Data Editor.

For information on how to use the Manual Data Editor , see:

- Editing existing logged measurement data
- Adding manually collected measurement data
- Viewing previous change details
- Reverting previous data edits
- Creating new measurements

For reference information see:

- Manual Data Editor user interface

Viewing previous change details

Manual Data Editor maintains an audit log which records all changes to logged data made by the tool. The log is filtered by source/measurement.

To view previous change details:

1. In Manual Data Editor, click **Load New Data** to open the Load Data dialog box.

   NOTE: Load Data opens automatically when you start Manual Data Editor.

2. In Load Data:

   - Select the Source for which you want to see the audit log.

   - Select the Measurement for which you want to see the audit log.

   - Click **Add Data to Selection** to add the source/measurement to the list of items to be opened in the data editor.

3. (Optional) Repeat step 2 to add additional sources and measurements to the selection list.

4. Click **Load Data** to open the source/measurement data entry fields in the data editor.

   NOTE: Ignore the settings for **Time Range** and **Options**, they have no influence on the display of the audit log.

5. (Optional) In the data editor, click **Load New Data** to change which data is loaded, the time range, or the editor options.

6. Click **View Log** to open the audit log for these sources/measurements.

7. View the audit log.

8. Close Manual Data Editor.

For information on how to use the Manual Data Editor , see:

- [Editing existing logged measurement data](#)
- [Adding manually collected measurement data](#)
- Viewing previous change details
- [Reverting previous data edits](#)
- [Creating new measurements](#)

For reference information see:

- [Manual Data Editor user interface](#)

Reverting previous data edits

It is not possible to directly and automatically revert previous data edits with Manual Data Editor. However, you can use the information available in the audit log to manually revert previous edits.

To revert previous data edits:

1. In Manual Data Editor, click **Load New Data** to open the Load Data dialog box.

   NOTE: Load Data opens automatically when you start Manual Data Editor.

2. In Load Data:
   - Select the Source for which you want to revert previous edits.
   - Select the Measurement for which you want to revert previous edits.
   - Click **Add Data to Selection** to add the source/measurement to the list of items to be opened in the data editor.

3. (Optional) Repeat step 2 to add additional sources and measurements to the selection list.

4. In Load Data, select the **Time Range** for the logged data you want to open in the data editor.

5. (Optional) In Load Data, set the following editor options:
   - **Timezone** - Set the timezone for the display of the configurable timestamp in the data editor. The data editor shows two timestamps for each data record, one timestamp in UTC and one in a configurable timezone.

- **Timestamp Format** - Set the format for the timestamp display in the data editor.

- **Number Format** - Set the number of decimals that are displayed for the logged data in the data editor.

- **Is Interval Data** - Select the check box to enable this property. Set the expected logging interval for the data.

  This setting ensures that there is a data row in the editor for every expected log entry. If there is no data record for an expected entry in the database, then the Manual Data Editor inserts a timestamped row with a null value in the editor. This makes it easier to identify missing records and correct them.

  NOTE: The data editor shows all logged records in the selected time range, regardless of the Interval Data settings. No records are hidden or filtered out.

6. Click **Load Data** to open the source/measurement data in the data editor.

7. (Optional) Click **Load New Data** to change which data is loaded, the time range, or the editor options.

8. Click **View Log** to open the audit log for these sources/measurements.

9. Find the log entries that you want to revert. Using the information provided in the **Original Value** and **New Value** columns, identify the values you want the logs to be reset to.

   TIP: You can copy the values from the audit log and paste them into the data editor or an external editor such as Microsoft Excel.

10. **Close** the Audit Log.

11. In the data editor, locate data records that you want to edit and use any of the following tools or methods to reset the data:

    - Click a data record and edit the data value.

    - **Insert Timestamp** - Insert a new data record with a timestamp that you specify. The timestamp must be in the time range of the data that is loaded into the editor. The new record has <no data> data values.

      NOTE: A historical data log timestamp marks the end of the logging interval. For example, a data log with a 15 minute logging interval and timestamp of 17:00 represents data for the time interval of 16:45 - 17:00.

    - Delete a data record or data value. These options are available in the right-click context menu.

    - Set a data value to <null>. This option is available in the right-click context menu.

    - Paste the data values from the audit log into the data records. The copy and paste options are available in the right-click context menu.

      NOTE: Only select the data values for copy/paste, not the timestamps.

    TIP: Modified data values are shown in bold font in the Manual Data Editor.

12. (Optional) In the data editor, use **Locate Changes** to find and step through the edited data records.

13. (Optional) Revert edited data values back to their original value. This option is available in the right-click context menu. Use **Undo all Changes** to revert all edits in the loaded data set.

14. Click **Apply Changes** to write the data log changes to the database.

15. Close Manual Data Editor.

For information on how to use the Manual Data Editor , see:

- [Editing existing logged measurement data](#)
- [Adding manually collected measurement data](#)
- [Viewing previous change details](#)
- Reverting previous data edits
- [Creating new measurements](#)

For reference information see:

- [Manual Data Editor user interface](#)

Creating new measurements

You can use Manual Data Editor to create new, custom measurements in WinPM.Net. These measurements are available anywhere in the software.

To create a new measurement:

1. In Manual Data Editor, click **Load New Data** to open the Load Data dialog box.

   NOTE: Load Data opens automatically when you start Manual Data Editor.

2. In Load Data, click **Create new Measurement** to open the Create new Measurement dialog box.

   NOTE: Check the list of available measurements to see if one exists for your application before creating a new measurement.

3. In Create new Measurement, enter the relevant information for the new measurement.

4. Click **Create** to add the new measurement to the system.

5. (Optional) Repeat steps 2-4 to add additional measurements.

6. Close Manual Data Editor.

For information on how to use the Manual Data Editor , see:

- [Editing existing logged measurement data](#)
- [Adding manually collected measurement data](#)
- [Viewing previous change details](#)

- [Reverting previous data edits](#)

- Creating new measurements

For reference information see:

- [Manual Data Editor user interface](#)

PQDIF Exporter

PQDIF (Power Quality Data Interchange Format) is a non-proprietary means of exchanging power quality data between different metering devices and software.

Developed under the guidelines of IEEE P1159.3, PQDIF provides a common ground where different vendors can export to or import from, using a data format consistent with the defined PQDIF standard.

WinPM.Net supports PQDIF through the "PQDIF Exporter" add-on. PQDIF Exporter takes data from the WinPM.Net SQL Server database, converts it to PQDIF, and stores this formatted file for viewing with any of the various programs that can display PQDIF data. The PQDIF Exporter allows you to set regularly scheduled exports of WinPM.Net data to PQDIF, as well as perform these exports manually.

This guide provides basic configuration and operation instructions for the PQDIF Exporter. It is assumed that the user already has knowledge of PQDIF. For details on viewing the PQDIF data, refer to the documentation that came with your PQDIF viewer. For detailed information on the PQDIF specification, visit the IEEE website.

Configuring the PQDIF Exporter

From Management Console, select **Tools > System > PQDIF Exporter**.

When you run PQDIF Exporter for the first time, the PQDIF Exporter Configuration dialog opens with the options to perform the initial configuration steps. To edit these settings at any time, select **Options > Configure PQDIF Exporter**. You can also edit the PT/CT Scaling used for the sources by the PQDIF Exporter; select **Options > PT/CT Scaling**.
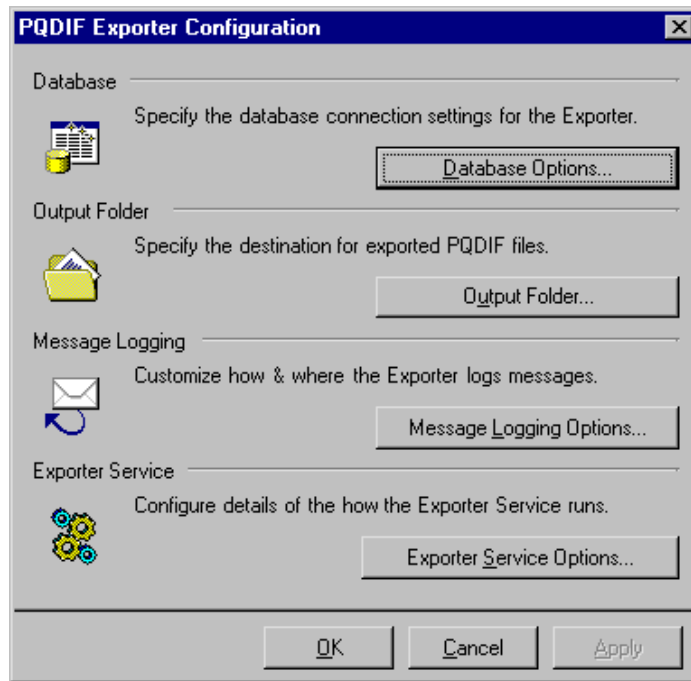
Database options

The Database Options dialog allows you to make changes to the database connection settings (SQL Server Instance, User Name, and Password). Type your changes in the appropriate fields.

NOTE: Under most conditions, you do not need to change the factory settings.

Click **Test These Settings** to test your database connection. A prompt displays to indicate whether or not connection to the database was successful.

Output folder

Click **Output Folder** to change the folder where PQDIF Exporter stores its output files. Type the folder path or click **Browse** to navigate and select the folder (or to create a new folder).

Message logging options

Click **Message Logging Options** to change the settings for message logging and email notification. The options available to configure are Log File, System Log, and E-mail Notification.

Log File options

Click **Log File Options** to specify the location for the log file and the logging level option.

Location

To change the folder location for the log file, type the folder path in the Location field or click **Browse** to navigate and select the folder.

Logging Level

Logging Level lists the different options for logging depth. Select the level you want from the dropdown list:

- **Disabled**: No messages are stored in the log file.

- **Normal**: Warning and error messages are stored in the log file.

- **Detailed**: Warnings, errors, and information are stored in the log file.

- **Diagnostic**: Warnings, errors, information, and diagnostic messages are stored in the log file.

NOTE: The "Diagnostic" log file option uses a huge amount of system resources. Select this option only if absolutely necessary.

System Log options

Click **System Log Options** to specify whether or not messages are stored in the WinPM.Net system log.

Select from the following options:

- **Disabled**: No messages are stored in the system log.

- **Normal**: Warning and error messages are stored in the system log.

Email options

Click **E-mail Options** to set up email notification from PQDIF Exporter.
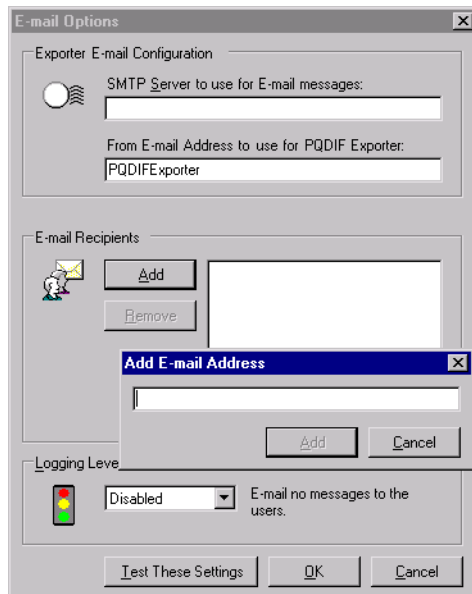
SMTP Server to set for E-mail messages

Type the name of the SMTP server PQDIF Exporter uses to send the email.

From E-mail Address to use for PQDIF Exporter

Type the address you want to appear in the "From" field in the email message.

E-mail Recipients

Click **Add** to enter email address recipients for the PQDIF Exporter messages. To delete an email address from the list, select it then click **Remove**.

Logging Level

The Logging Level box lists the different options for the amount of information sent in the email message. If you change this setting, it affects the setting for all email recipients.

- **Disabled**: No email sent.

- **Normal**: Email warning and error messages.

When you have finished setting up the email options, click **Test These Settings** to test the email connection.

If email settings are configured properly, a test message is sent to your inbox.

Exporter service options

Click **Exporter Service Options** to specify how often the PQDIF Exporter service restarts from an idle state to check the database for data to export.

Use the up and down arrows and dropdown list to set the frequency.

> **NOTE**: Setting the Exporter Service Frequency Settings to seconds uses a huge amount of system resources. Select this option only if absolutely necessary.

PT/CT scaling

For a network where the PT/CT ratios on the meters are not set to the actual ratio values (for example, if the ratios are set to 1:1 for revenue metering purposes), the PQDIF Exporter service can scale the values to the correct primary measurements when it exports data.

To set the PT/CT scaling for the PQDIF Exporter:

1. Click **Options > PT/CT Scaling** to open the PT/CT Scaling dialog displaying the current PT/CT scaling for the sources in your network.

2. Select a source(s) and click **Edit PT/CT Scaling for Selected Sources** to open the Edit PT/CT Ratios dialog.

3. Enter the multiplier required to convert secondary values to primary values (for example, enter 1000 for a 5000:5 CT). Click **OK**.

4. Click **Save** to save your changes.

PQDIF Exporter main window

This is the main user interface for setting up schedules for PQDIF Exporter jobs or for setting up and running the jobs manually.

From Management Console, click **Tools > System > PQDIF Exporter**.

	A	Main menu items	B	Status of Exporter
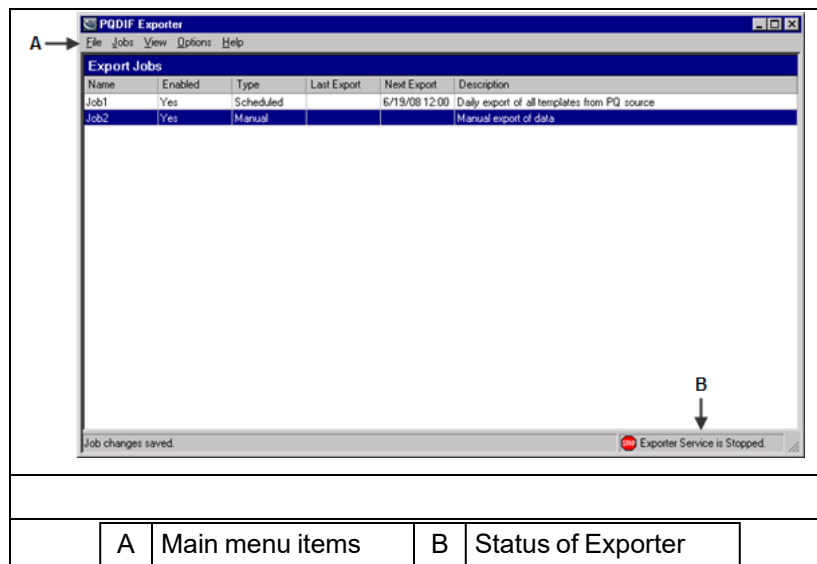
Menu items

All the PQDIF Exporter commands are available from the main menu. Some of these commands are also available when you right-click on the interface background or on a job.

Starting/stopping the exporter service

To start the PQDIF Exporter service:

* Select **Options > Start Exporter Service**, or

* Right-click the status bar and select Start Service.

To stop the service:

* Select **Options > Stop Exporter Service**, or

* Right-click the status bar and select Stop Service.

PQDIF Exporter jobs

The **Jobs** menu contains commands which allow you to set up PQDIF Exporter jobs that run automatically, according to the schedule you specify. You can also set up manual jobs to collect data for a specific date range from the SQL Server database for conversion to PQDIF.

PQDIF data templates

PQDIF data templates are the mapping files that convert WinPM.Net data to PQDIF. PQDIF Exporter extracts recorded data for the following default templates:

* **Flicker**: Short-term and long-term flicker disturbance data on the voltage inputs.

* **Sag/Swell**: Sag/swell disturbance data for the voltage inputs, including minimum, maximum and average values.

* **Sag/Swell Waveforms**: Waveform data for voltage sag/swell.

* **Steady-state**: Steady-state (RMS) data for trending.

* **Steady-state Waveforms**: Waveform data for steady-state data.

> **NOTE**: For the Steady-state Waveforms template, you must program the meter to use a Periodic Timer module to perform periodic waveform recording of the voltage and current channel inputs. The time interval you specify must be a multiple of 15 minutes.

- **Transient Waveforms**: Transient disturbance data for voltage inputs.

In addition to the default templates, custom templates can be also be developed — contact Technical Support if you require customized data templates for your application.

Scheduled export jobs

You can set up PQDIF Exporter to run at a specific time every day, or you can specify a different time interval (for example, every 12 hours or every 7 days). You can create and store multiple scheduled jobs for the PQDIF Exporter.

Creating a Scheduled Export Job

1. Click **Jobs > New Job > Scheduled Export** to open the **Scheduled Export** dialog.
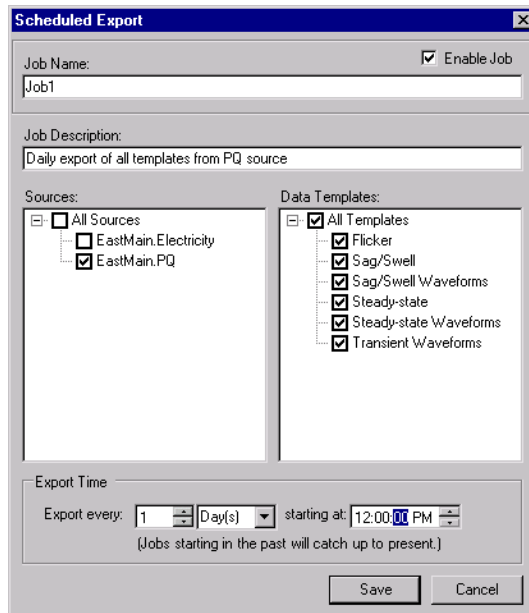
2. Type a short but descriptive name in the **Job Name** field. Use the **Job Description** field to give more details about the job.

3. Select the sources for your scheduled data export. Click the box beside each source to select or deselect it.

   Click **All Sources** to select all available devices that log the type of data specified by the data templates. When this logged data is detected for a new device, that device is automatically added to all scheduled PQDIF Exporter jobs that have All Sources selected.

4. Select the Data Templates you want to use when the PQDIF Exporter runs. Click the box beside each template to select or deselect it.

   Click **All Templates** to select all available data templates for the selected devices. When a new data template is added, that template is automatically added to all scheduled PQDIF Exporter jobs that have All Templates selected.

5. Use the **Export Time** section to specify how frequently the PQDIF Exporter should run, as well as its start/cut-off time. When a scheduled PQDIF Exporter job runs for the first time (at the specified start time), it collects historical data to create PQDIF files so that the time period for that export is consistent ("caught up") with the time periods of all subsequent scheduled PQDIF exports.

  - **Example 1**: If you saved the scheduled job (1) at 4:25 PM, with the Exporter scheduled to run daily at midnight, the first PQDIF file is generated at midnight (2) the same day, and includes data spanning from the previous midnight. PQDIF Exporter runs again next midnight (3), and includes data spanning the first scheduled export (2).
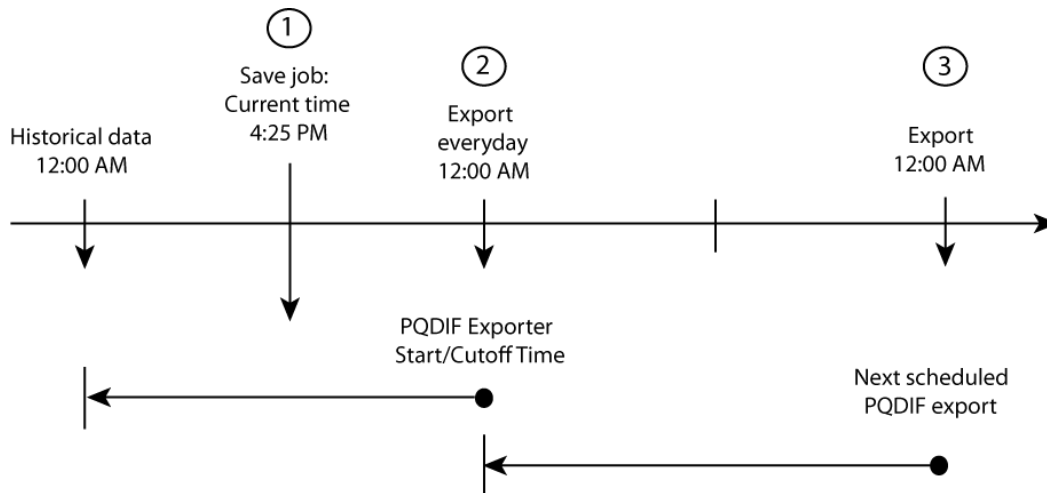
  - **Example 2**: If you set a scheduled job to run PQDIF Exporter every 8 hours, starting at 3:00 AM, then the first PQDIF file is generated at 3:00 AM, and will include data spanning 8 hours previous to that time (that is, 7:00 PM). The next export starts 8 hours later, at 11:00 AM, then at 7:00 PM, then at 3:00 AM and so on.

  - **Example 3**: To run PQDIF Exporter once a week at 6:00 AM, change Export Time to "Export every 7 Day(s) at 6:00 AM". In this example, the PQDIF file contains a full week's worth of PQDIF data

6. Click **Add Job**.

NOTE: Exporter Service must be started in order to run scheduled PQDIF Exporter jobs. See PQDIF Exporter main window.

Manual export jobs

A manual export job enables you to run the PQDIF Exporter immediately. You can create a different manual job for each source and/or template, or for specific groups of sources and/or templates.

Creating a manual export job

1. Click **Jobs > New Job > Manual Export** to open the **Manual Data Export** dialog.

2. Type a short but descriptive name in the **Job Name** field. Use the **Job Description** field to give more details about the job.

3. Select the sources for your data export. Click the box beside each source to select or deselect it.

   – To select all available devices, select the **All Sources** box.

4. Select the Data Templates you want to use. Click the box beside each template to select or deselect it.

   – To select all available templates, select the **All Templates** box.

5. Use the Export Range **From** and **To** boxes to specify the date range of historical data that the PQDIF Exporter will use. The default time format is local time. If you want to use Coordinated Universal Time format, select **UTC**.

6. Use the File Splitting options to specify how PQDIF Exporter exports the data. The default method is to export the data to a single PQDIF file. You can change this so the data is split into different PQDIF files — for example, if you want to split the data so each PQDIF file contains 12 hours worth of data, set the File Splitting to "Export into separate files for each 12 Hour(s) of data".

7. Click **Export Now** to run and save this manual PQDIF Exporter job, or click **Add Job** to save without running it.

Running manual jobs

After you enter the manual PQDIF Exporter jobs, you can select and run them:

1. On the PQDIF Exporter interface, select the manual job(s) you want to run.

2. Right-click and select **Export Selected Manual Job(s)**. A command window displays to indicate the progress of the export. Press the RETURN (ENTER) key when prompted to return to the PQDIF Exporter interface.

> **NOTE**: To run all manual jobs, click **Jobs > Export All Manual Jobs**.

Editing the PQDIF Exporter jobs

To make changes to a scheduled or manual job:

1. To edit a job, open the **Export Configuration** dialog in one of the following ways:
   - Double-click the job.
   - Select the job, then click **Jobs > Edit Selected Job**, or
   - Right-click the job and select **Edit Selected Job**.

2. Make your changes and click **Save**.

Duplicating PQDIF Exporter jobs

If you are creating jobs that have very similar attributes (for example, jobs that use the same set of devices, templates and/or time schedules), you can set up the first job, duplicate it, then simply edit the copy.

1. Select the job(s) you want to duplicate, then click **Jobs > Duplicate Selected Jobs** (or right-click the job and select **Duplicate Selected Jobs**).

2. Double-click the copy of the job you want to edit, make your changes, then click **Save**.

Enabling/disabling the PQDIF Exporter jobs

To enable or disable the scheduled or manual jobs:

1. Select the job(s) you want to enable or disable.

2. To enable the jobs, click **Jobs > Enable Selected Jobs**. To disable them, click **Jobs > Disable Selected Jobs**. You can also right-click the jobs and select **Enable Selected Jobs** or **Disable Selected Jobs** to enable or disable the jobs.

Adjusting the column widths

To change the width of a column, hover the mouse over border of the column headings — the mouse pointer changes into a double-arrow pointer — click and drag to adjust the column width.

Sorting the PQDIF Exporter job list

Click the title of a column to sort the data according to that column heading. An arrow appears in the title of the column. Click the title again to toggle the sorting—ascending or descending order.

Deleting PQDIF Exporter jobs

To permanently remove PQDIF Exporter jobs from the list:

1. Select the jobs you want to delete, then:

    −   Click **Jobs > Delete**,

    −   Press the Delete key on your keyboard, or

    −   Right-click and select **Delete Selected Jobs**.

2. Click **Yes** when prompted.

> **NOTE**: You can also disable a job instead of deleting it (that is, Enable = "No"). This way, the job can be enabled at a later date if required.

Closing the PQDIF Exporter window

To close and exit the PQDIF Exporter window, click the "X" at the top right corner, or click **File > Exit**.

If the Exporter Service is stopped when you try to exit, PQDIF Exporter prompts you to start the service in order to process scheduled jobs. Click **Yes** to start the Exporter Service.

Remote Modem Setup

Use Remote Modem Setup to preconfigure dialup modems that will be connected to meters at remote locations. Hardware-based and software-based (WinModems) modems are supported.

Remote Modem Setup provides access to a database that contains configuration data for a variety of modems you can pick from. Remote Modem Setup also provides direct access to your modem so you can customize settings and troubleshoot its operation.

Preliminary Setup

Follow the instructions in your modem's documentation to set up and connect the modem to one of your computer's serial ports. Select a port that the WinPM.Net Communications Server is not using.

Using Remote Modem Setup

1. Start Management Console.

2. Click **Tools > System > Remote Modem Setup** to open the **Remote Modem Setup** dialog.

3. Set the Port to the serial communications port that the computer uses to communicate with (and set up) the modem. Set the Baud Rate to match the modem's baud rate.

    > **NOTE**: To reduce possible communication issues, set the computer, modems and meters to the same baud rate.

4. Set the following properties for the remote modem:

    −   Select the type of modem from the **Type** box.

    −   Set the Site Baud Rate.

- To set the modem to answer after a specified number of rings, select **Auto Answer**, then type the appropriate number in the Number of Rings box.

- To save the modem configuration data in the modem's onboard memory, select **Store Profile**. This feature allows the modem to revert to the saved settings after a power outage. Clear **Store Profile** to disable this feature.

- To turn on the modem's internal speaker, select **Speaker On**, then select a setting from the Volume list. Clear **Speaker On** to disable this feature.

5. Click **Send** to save the settings to the modem.

   If there are DIP switch settings required for the modem that is being configured, a message displays with instructions on which switches to turn on or off. Set the DIP switches on the modem as instructed.

The modem is now ready to be used at the remote power monitoring site.

Reports Configuration

Before you can use Reports, you need to specify the database to use for reporting, and you need to configure the delivery options for report subscriptions. These settings are defined in Management Console.

To set the prerequisites:

1. Open Management Console and select **Tools > Reports Configuration** to open the dialog.

2. Click the **Reports** tab to configure general reporting options:
   - From the dropdown list in the **ReportsDatabase** section, select the database on which to run reports. Only those databases and database archives compatible with Reports are listed.
   - (Optional) Use the slider in the **Report Timeout** section to set the time (in seconds) after which Reports stops trying to complete a report generation task.

3. Click the **Subscriptions** tab to configure the settings for report subscriptions:
   - **SMTP Server**: Enter the IP address of your SMTP server, or the fully-qualified network server name from which the report subscriptions are emailed.
   - **Port**: Type the port address used by your SMTP server.
   - **Use SSL**: Select this option to encrypt your user credentials.
   - **Username**: If required by your SMTP server, enter the Windows user name used to configure the SMTP server settings.
   - **Password**: If required by your SMTP server, enter the password associated with the Windows user name.
   - **"From" Display Name**: (Optional) Type the name that you want to appear in the "From" field of an emailed report.
   - **"From" Email Address**: Type the email address that you want to appear in the "From" field of the emailed report.

4. Click **OK** to save your changes.

Setting Reports prerequisites

Before you can use Reports, you need to specify the database to use for reporting, and you need to configure the delivery options for report subscriptions. These settings are defined in Management Console.

To set the prerequisites:

1. Open Management Console and select **Tools > Reports Configuration** to open the dialog.

2. Click the **Reports** tab to configure general reporting options:

    - From the dropdown list in the **Reports Database** section, select the database on which to run reports. Only those databases and database archives compatible with Reports are listed.

    - (Optional) Use the slider in the **Report Timeout** section to set the time (in seconds) after which Reports stops trying to complete a report generation task.

3. Click the **Subscriptions** tab to configure the settings for report subscriptions:

    - **SMTP Server**: Enter the IP address of your SMTP server, or the fully-qualified network server name from which the report subscriptions are emailed.

    - **Port**: Type the port address used by your SMTP server.

    - **Use SSL**: Select this option to encrypt your user credentials.

    - **Username**: If required by your SMTP server, enter the Windows user name used to configure the SMTP server settings.

    - **Password**: If required by your SMTP server, enter the password associated with the Windows user name.

    - **"From" Display Name**: (Optional) Type the name that you want to appear in the "From" field of an emailed report.

    - **"From" Email Address**: Type the email address that you want to appear in the "From" field of the emailed report.

4. Click **OK** to save your changes.

Software Logging

Use Software Logging to add or edit software-based data logging for Modbus device types. This is intended for device types that do not have onboard data logging capabilities. You can also add software logging to a device type with onboard data logging to log additional measurements that are not logged by the device.

RECOMMENDATION: Use onboard data logging when possible. The data log timestamps for onboard data logs are more accurate than the timestamps for software data logging. Onboard data logging is also more reliable than software data logging since it does not depend on a network connection to the device.

The **Software Logging** link in Management Console opens the same **Configure Logging and Calculation** dialog that is available through the Device Type Editor. For more information on adding software logging see the Device Type Editor Help > Adding Software Logging to a Modbus Device Type.

Time of Use Editor

Time of Use (TOU) schedules created with the Time of Use Editor are designed to be used with the Reports application. When you use a TOU schedule in a report, the report data is presented according to the time periods you defined in the schedule. See *Report definitions* in the online help for Reports in the Web Applications component for information on the Energy Cost Report.

A typical TOU has On Peak, Off Peak and Shoulder periods. Since rate schedules vary not only by day and week, but also by season, the TOU schedules can be set up to match your precise needs.

Opening the Time of Use Editor

To access the Time of Use Editor, open Management Console and select **Tools > System > Time of Use Editor**.

Use the navigation pane on the left to navigate through the schedules. Right-click a schedule or sub-schedule to access menu items. Select an item to display associated tabs in the tab display pane on the right.

Overview: Creating a TOU schedule

To create a TOU schedule, you need to:

1. Add a schedule. See Adding or editing a schedule.

2. Add a sub-schedule, if you need more than one sub-schedule. See Adding or editing a sub-schedule.

3. Add a season, if you need more than one. See Configuring seasons.

4. Configure special days (such as holidays) to include in your TOU schedule. See Adding special days.

5. Add day groups, if you need groups other than the default day groups. See Configuring Day groups.

6. Add TOU names, if you need names other than the default names. See Defining TOU names.

7. Assign TOU periods to TOU names. See Defining TOU periods.

To view a sample schedule, right-click the navigation pane and select **Import Schedule**. Select SampleSchedule.tou from the ...\WinPM.Net\system\etc folder and click **Open**.

Adding or editing a schedule

This section describes how to add a blank schedule, and import or copy an existing schedule.

Adding a blank schedule

1. Select **File > New Schedule** to open the **Add a Schedule** dialog.

2. Type a name and description for the schedule. Click the arrow beside the effective date to bring up a calendar where you can select the effective date of the schedule.

3. Click **OK**. That schedule name appears in the list in the navigation pane.

Copying an existing schedule

1. In the navigation pane, right-click the schedule that you want to copy.

2. Select **Save as New Schedule** to open the **Save as New Schedule** dialog.

3. Enter a name and description for the new schedule.

4. Click **OK**. The new schedule appears in the navigation pane with the attributes of the copied schedule.

Importing a schedule

1. Place the schedule that you want to import into a folder that can be accessed from the computer.

2. In the navigation pane, right-click an empty area of the navigation pane then select **Import Schedule** to open the **Open** dialog.

3. Navigate to the folder where you placed the schedule. Select the schedule and click **Open**.

   The schedule is imported with the extension "_imported". For example, a schedule named "ScheduleA" becomes "ScheduleA_imported".

Exporting a schedule

To export a schedule to use in another TOU installation:

1. Right-click the schedule you want to export in the navigation pane. Click **Export** to open the **Save As** dialog.

2. Select the folder where you want to save the schedule.

3. Enter a schedule name in the File name box.

4. Click **Save**.

   The schedule is saved with the extension ".tou". For example, "ScheduleB" becomes "ScheduleB.tou".

Editing an existing schedule

To edit a schedule name or description:

1. From the navigation pane, right-click a schedule name then click **Edit Schedule** to open the **Edit a Schedule** dialog.

2. Change the schedule name or description.

   To change the effective date, you must make the change to the sub-schedule(s). See [Editing a sub-schedule effective date](#).

3. Click **OK**.

Deleting a schedule

To delete a schedule, right-click the schedule name in the navigation pane then select **Delete Schedule**. This deletes the entire schedule, including any sub-schedules.

Adding or editing a sub-schedule

When you add the basic schedule, it contains one sub-schedule, with the effective date (start date) you entered for the schedule. To add additional sub-schedules with alternate effective dates, follow the instructions in the following sections.

Expand a schedule to display its sub-schedule(s). Sub-schedules are listed in order of their effective dates.

TIP: If you want to have more than one sub-schedule with the same seasons, day groups, TOU periods, etc., configure the first sub-schedule then right-click it and select **Save as New SubSchedule**.

Adding a blank sub-schedule

1. Right-click the schedule name and select **New SubSchedule** to open the **New SubSchedule** dialog.

2. Click the arrow beside the effective date to open a calendar where you can choose an effective date for the new sub-schedule.

3. Click **OK**.

   The sub-schedule displays in the navigation pane. Continue configuring the sub-schedule by adding seasons and day groups.

To add a new sub-schedule that has the same settings as an existing sub-schedule, see Copying an existing sub-schedule, below.

Copying an existing sub-schedule

1. Right-click the sub-schedule that you want to copy and select **Save as New SubSchedule** to open the **New SubSchedule** dialog.

2. Click the arrow beside the effective date to open a calendar where you can choose an effective date for the new sub-schedule.

3. Click **OK**. The new sub-schedule displays in the navigation pane with the same attributes (other than effective date) of the sub-schedule you copied.

Editing a sub-schedule effective date

1. Right-click the sub-schedule for which you want to change the effective date, then select **Edit Effective Date** to open the **Edit SubSchedule Effective Date** dialog.

2. Click the arrow beside the effective date to open a calendar where you can choose an effective date for the new sub-schedule.

3. Click **OK**.

Deleting a sub-schedule

To delete a sub-schedule, right-click the sub-schedule name in the navigation pane and select **Delete**. This also deletes all seasons in that sub-schedule (and any day groups, special days, etc., that they contain).

Configuring seasons

Use seasons to create timeframes for presenting data in reports. A newly created schedule has a single season, called All Year. Expand a sub-schedule to view its seasons.

If you only need one season (All Year), you can skip the steps below and go to configuring special days.

Configuring seasons

Use the **Define a Season** tab to set up or edit the time spans for each season.

> **NOTE**: A date can only belong to one season; you cannot overlap dates in one season with dates in another.

1. From the navigation pane, select the sub-schedule to which the season belongs.

2. In the tab display pane, select the **Define a Season** tab.

   After you add a sub-schedule, the tab displays only one season (All Year) that covers the entire year. The season is listed at the top of the tab, and the calendar that shows the days and months that belong to that season is full. To make room to add other seasons, you must edit the All Year season.

3. Select **All Year** from the **Define a Season** tab and click **Edit** to open the **Edit a Season** dialog.

4. In the **Season Name** field, type the name of one of the seasons you want to define (for example, Summer).

5. Set the Start Date and End Date for the season. You can specify fixed start and end dates (For example, 12/31) or relative start and end dates (for example, the last Saturday of October).

   Fixed Date: Click on the month or day portion of the date then either type the new date or use the up and down arrows beside the date to edit it.

   Relative Date: Select the day and month from the dropdown lists.

   For the end date, select the Time of Day the season ends (Start of Day or End of Day). If you select Start of Day, the season ends at 12:00:00 AM on the selected day; if you select End of Day, the season ends at 12:00:00 AM on the next day so there are no gaps in the schedule.

   > **NOTE**: If you use relative dates, you can configure a schedule that may become invalid in future years. Be careful when you configure the schedule that there will be no missing or overlapping days in the schedule in future years. For example, rather than scheduling one season to end on the 1st Sunday in June and the next to start on the 1st Monday in June, it is better to schedule the first season to end at Start of Day on the 1st Sunday in June and the next season to start on the 1st Sunday in June.

6. If this is a split season (where there is a gap between two parts of the season), select the Split Season check box and add a second sets of dates. For example, winter is typically a split season because it occurs in the first and last months of the year.

7. Click **OK**.

   The calendar on the **Define a Season** tab now shows one season with the rest of the year unassigned.

8. Click **Add** to create additional seasons. Repeat steps 4 to 6 to name the season and define the start and end dates.

   NOTE: Do not leave any dates unassigned: unassigned dates are not calculated in the reports.

Editing a season

1. On the **Define a Season** tab, select the season you want to edit then click **Edit** to open the **Edit a Season** dialog.

2. Make the changes you require then click **OK**.

Deleting a season

To delete a season, select the season in the **Define a Season** tab then click **Delete**. This also deletes the day groups and TOU periods configured for that season.

Adding special days

After you define a sub-schedule, you need to set up holidays and other special days. You can define holidays that occur on the same date every year or at some other regular interval (such as the third Thursday in November). You can also define special days, such as inventory days, that might occur regularly and require special treatment in reports.

Use the **Define Special Days** tab to set up holidays and other special days.

1. In the navigation pane, select the sub-schedule for which you want to define special days.

2. In the tab display pane, select the **Define Special Days** tab.

3. Click **Add** to open the **Add a Special Day** dialog.

4. Type a name for the special day in the Name field.

5. Select the frequency of the day:
   - **One time only**: Enter the date for a single day that does not repeat in future years; for example, a special day for your organization that does not repeat every year.
   - **Repeat yearly**: Enter the date; for example, a holiday that occurs on the same date every year.
   - **Repeat on**: Complete the fields in the Repeat On section to define how the day repeats and in which months it occurs. Select whether it occurs on a particular date of the selected month(s) (Day) or a relative day (i.e., Last Saturday) in the selected month(s).

6. Repeat steps 3 to 5 to add additional special days.

7. Click **OK** to save your changes and return to the **Define Special Days** tab.

See [Configuring Day groups](#) for instructions on how to apply special days to a sub-schedule.

Deleting a special day

To delete a special day, select the special day in the **Define a Special Day** tab then click **Delete**.

Configuring Day groups

Day groups define how days of the week are divided into groups.

Day groups can include:

- days of the week

- special days, such as holidays or other specially designated days (such as inventory days)

Default settings are weekdays (Monday to Friday) and weekends (Saturday and Sunday) but you can add additional groups. There can be a maximum of 47 day groups in any season.

Make sure that all days have been accounted for since unaccounted time is not included in any reports.

Adding a Day group

Before you add a day group, make sure that you have defined seasons and special days for the sub-schedule.

1. In the navigation pane, select the season for which you want to define day groups. The **Define DayGroups** tab appears in the tab display pane.

2. Click **Add** to open the **Add a DayGroup** dialog.

   If the Add button is unavailable, there are no free days to add to a day group. A day can only belong to one day group. To add another day group, you must first remove days from an existing day group.

3. Type a name for the day group.

4. Select one of the following options:

   - **Day of week**: Select the check box(es) for the day(s) you want to include. A day can belong to only one day group; if a day is unavailable, it already belongs to an existing group. If all days are unavailable, you can only include special days in this group unless you remove days from another day group.

   - **Special Day(s) Only**: Select the special day in the Include Special Days list then click to select or clear the day. Days with a check mark beside them are included in the day group. You can only include a special day in one day group.

5. Click **OK**. The name is added to the day group list.

Editing a Day group

1. On the **Define DayGroups** tab, select the day group you want to change, then click **Edit** to open the **Edit a DayGroup** dialog.

2. Make the changes you require then click **OK**.

Deleting a Day group

To delete a day group, select the day group in the **Define DayGroups** tab then click **Delete**. When you delete a day group, you also delete the TOU periods configured for that day group. If you add a day group again, you will need to reconfigure the TOU periods.

Configuring TOU names and TOU periods

When you run reports, the reports calculate and display data according to the information you enter in the **Define TOU Periods** and **Define TOU Names** tabs.

NOTE: You need to define TOU names and periods for each schedule that you create. Be sure to apply a TOU name to every square on the calendar. Unaccounted time causes your reports to display inaccurate information.

Defining TOU names

A TOU name is a designation that is used to differentiate billing rates. Three TOU Names are already defined: On Peak, Off Peak and Partial Peak. You can edit or delete these names, or add new names.

To define TOU names:

1. In the navigation pane, select the day group for which you want to define TOU names then select the **Define TOU Names** tab.
2. Click **Add** to open the **Add a TOU Name** dialog.
3. Type the TOU name you want to add then click **OK**. The TOU name is added to the list of TOU names.

Editing an existing TOU name

1. Select the TOU Name in the list on the **Define TOU Names** tab and click **Edit** to open the **Edit a TOU Name** dialog.
2. Type the new name in the TOU name field.
3. Click **OK**.

Deleting a TOU name

Select the TOU name in the list on the **Define TOU Names** tab and click **Delete**.

NOTE: If you delete a TOU name, the TOU periods assigned to the name are replaced with "Unaccounted". Deleting a TOU name in a particular schedule affects all sub-schedules in that schedule.

Defining TOU periods

A TOU period is the time span within the 24-hour period to which you apply a TOU name (such as 8:00 a.m. to 5:00 p.m. for On Peak). You must have TOU names defined before you can associate TOU periods with those names.

Two TOU names are already defined: On Peak and Off Peak. You can edit or delete these names, or add new names.

Adding TOU periods

Select a day groups to set up the TOU time periods (on-peak, off-peak, etc.) for that group.

1. In the navigation pane, select the day group for which you want to define a TOU period then select the **Define TOU Periods** tab.

2. Click **Add** to open the **Add a Time Period** dialog.

3. In the Start Time field, type the time that the period begins, or select the hours and minutes and use the up and down arrows to set the start time.

4. In the Stop Time field, type the time that the period begins, or select the hours and minutes and use the up and down arrows to set the stop time.

5. From the Time of Use Name dropdown list, select the TOU name to associate with the time period.

6. Click **OK**.

To add a split time period (i.e., a time period that spans from 19:00 to 6:00), you must add two separate time periods and associate them with the same time of use name.

If you want to expand a time period into time that already belongs to another period, you must first edit the other time period to clear the time.

Continue adding time periods until the 24 hour period is filled. Unaccounted time blocks cause your reports to display inaccurate information.

Editing TOU periods

1. Highlight the period you want to edit and click **Edit** to open the **Edit a Time Period** dialog.

2. Make the required changes then click **OK**.

Deleting TOU periods

To delete a period, highlight the period and click **Delete**.

Update OPC Server

Use this function to initialize or update the OPC measurement mappings in the system. This function is part of the OPC server capabilities of WinPM.Net. For information on how to map ION registers to OPC, see OPC measurement support. For information on how to map Modbus registers to OPC, see Device Type Editor Help.

Upload Report Template

Use Upload Report Template to upload reports that are not available by default in the software. You can upload single report template or multiple report templates. The multiple report templates together are called as Report Pack.

NOTE: Information on how to create custom Report Templates is not provided in this help. Contact your local Siemens Industry representative for information on custom Report Templates.

For information on how to use the Upload Report Template, see:

- [Uploading Single Report Template](#)
- [Uploading Multiple Report Template (Report Pack)](#)

Uploading Single Report Template

You can upload single report template to use it along with other report templates.

To upload single report template:

1. In Upload Report Template, select **Report Template(.rdl)**.

2. Click **Select File** to navigate to the location of the .rdl file.

3. Select the file and click **Open**.

4. Click **Upload**.

    The uploaded report is in ...\ WinPM.Net\web\ReportDepot\ION Reports\ION Single Reports

NOTE: After you uploaded the Report Template, you can access it, together with the default templates, in the Report Library.

For information on how to use the Upload Report Template, see:

- Uploading Single Report Template
- [Uploading Multiple Report Template (Report Pack)](#)

Uploading Multiple Report Template (Report Pack)

You can upload multiple report template (Report Pack) at a time as a .zip file to use it along with other report templates.

To upload multiple report template (Report Pack):

1. In Upload Report Template, select **Report Pack(.zip)**.

    NOTE: Make sure that multiple report templates are zipped as a single .zip file.

2. Click **Select File** to navigate to the location of the .zip file.

3. Select the file and click **Open**.

4. Click **Upload**.

    A progress bar displays the upload status.

    The uploaded report are unzipped and is in ...\ WinPM.Net\web\ReportDepot\ION Reports\ION Report Packs

NOTE: After you uploaded the Report Template, you can access it, together with the default templates, in the Report Library.

For information on how to use the Upload Report Template, see:

- [Uploading Single Report Template](#)
- Uploading Multiple Report Template (Report Pack)

User Manager

User Manager is a Web-based application that you use to configure users and user groups. WinPM.Net does not provide any pre-configured user accounts or user groups. One supervisor account is created with a user defined password during the installation of the software. Create user accounts and groups to meet your needs. User Manager also gives information about the Web Application user licenses for the system.

Open the User Manager application from the Configuration Tools section on the **Settings** page in the Web Applications banner. You can also open User Manager from the **Tools** menu in the Management Console.

See the User Manager online help for information on how to use this application.

Virtual Processor setup

> **TIP**: You can open Virtual Processor Setup in Management Console from the **Tools > System > ION Virtual Processor Setup** link.

The Virtual Processor (VIP) is the WinPM.Net service (WinPM.Net Virtual Processor Service) that provides coordinated data collection and aggregation, control, and mathematical analysis of power monitoring system data from groups of metering or similar intelligent devices. Use Virtual Processor Setup to configure a Modbus Slave port when using the VIP as a Modbus Slave device for exporting data to an external, 3rd-party Modbus Master. . Virtual Processor Setup can also be used to improve the performance of the WinPM.Net Virtual Processor Service by modifying its global operating parameters.

See Virtual Processor service (VIP) for application examples using the Virtual Processor.

Configuring the Modbus Slave port

You can configure the VIP to act as a Modbus Slave device to an external Modbus Master. The Modbus Master can read data from the VIP through the VIP Modbus Slave modules. See the *ION Reference* for information on how to configure Modbus Slave modules.

For the Modbus Master to be able to access the VIP, you need to configure a Modbus Slave Port. A Modbus Slave Port can be a serial port, for example COM1, COM2, and so on, or a Modbus TCP port.

To add a Modbus Slave Port:

1. Open Management Console and log on with a supervisor level user account.

2. Select **Tools** > **System** > **ION Virtual Processor Setup**. This opens the ION Virtual Processor Setup dialog box.

3. In ION Virtual Processor Setup, find the VIP for which you want to add a slave port. Double-click **Modbus Network** for this VIP. This opens the Network view.

4. Double-click **Slave Ports**. This shows any configured slave ports for this VIP. If no slave ports are configured, the list will be empty.

5. Click **Add Port**. This opens the VIP Setup Modbus Port/Site dialog box.

6. In VIP Setup Modbus Port/Site, select the Port you want to use. Click **OK**.

7. Double-click the newly added slave port and then double-click **Settings**. This opens the port settings.

8. Edit the port settings if required. For more information on port settings, see VIP Modbus Slave port settings

9. Click **Save**, then click **Exit**.

> **NOTE**: You must restart the VIP for the changes to take effect.

Done

Modifying the Global Parameters

Typically, you do not need to change the default Virtual Processor's **Global Parameters**. However, the settings in the following table can be modified to improve system performance. To view and select the settings, expand Global Parameters.

Global Parameter	Value Range	Description	Notes
Module Update Period	50 to 1000 ms	The interval at which ION modules inside the Virtual Processor are updated.	Frequent module updates contribute to the workstation's processor load. By increasing the Module Update Period, you reduce the number of times a module executes in a given time period.
Configuration Saver Period	10 to 600 seconds	The interval at which the Virtual Processor writes its configuration data to the workstation's hard disk.	Increasing the Configuration Saver Period significantly reduces the processor load if you have a large Virtual Processor configuration file.
Client Polling Period	500 to 300000 ms	The interval at which the Virtual Processor gets information from other nodes in the network (for example, another Virtual Processor or a device).	You can increase the period for less time sensitive Virtual Processor applications and reduce the period for more time sensitive applications.
Server Polling Period	200 to 1440000 ms	The interval at which the Virtual Processor responds to requests for information from client nodes such as Vista or the Log Inserter.	Reducing this value increases the rate at which these responses are sent; however, this can also increase network traffic.

Chapter 4: Vista

Vista is the WinPM.Net component that displays and controls your power monitoring system.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software to control time-critical functions.
- Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

For information on starting and logging on to Vista or ending your Vista session, see Getting started.

The Vista interface

The Vista interface consists of a main screen with a title bar, a menu bar, a toolbar, a workspace, a status bar and (in Edit mode) a toolbox.

For information on opening or generating a Vista diagram, see Generating, opening and closing a user diagram.

The following image illustrates the Vista interface with a user diagram showing real-time values.
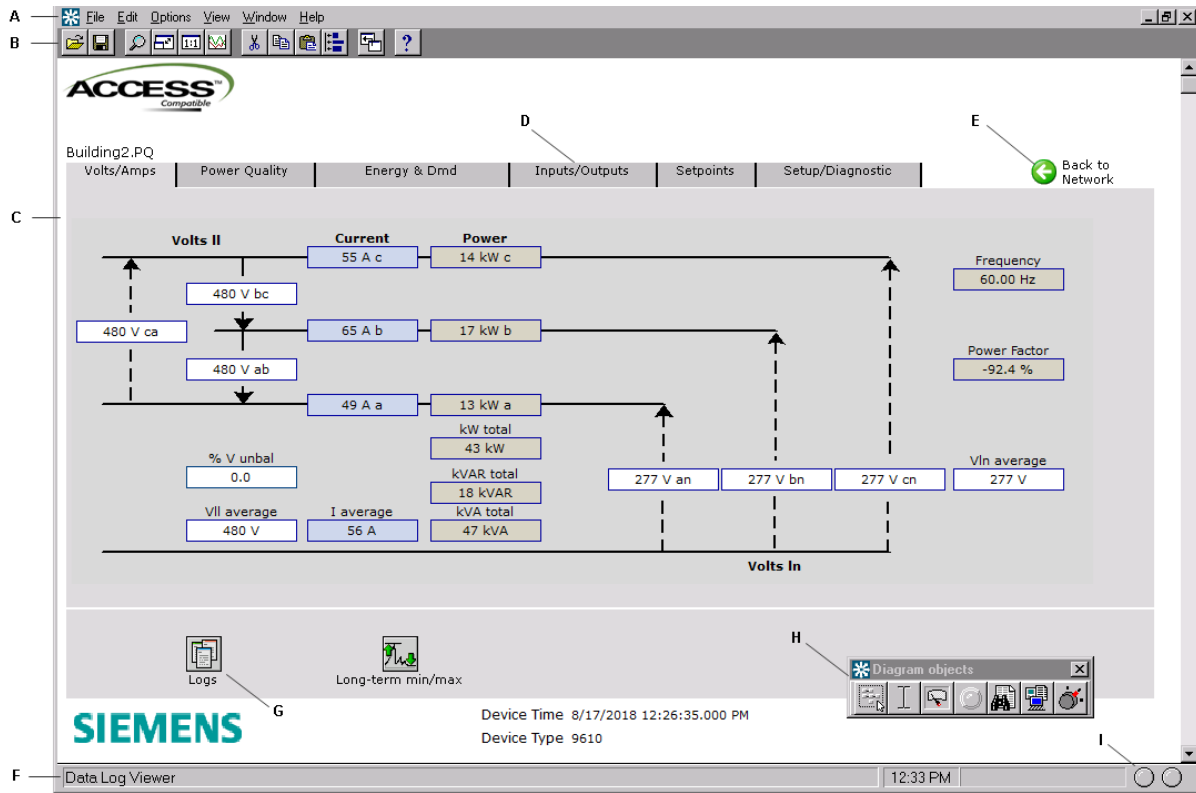
A	Menu Bar	B	Toolbar	C	Workspace
D	Click a tab to display its data	E	Click to return to the Network Diagram view	F	Status Bar
G	Click this grouping object to open another window containing more data	H	Toolbox	I	Receive/Transmit Status indicators

Toolbar

The toolbar offers quick access to the commands that are used most frequently. Each command on the toolbar is also available from the menus.

A	Open		G	Cut
B	Save		H	Copy
C	Zoom		I	Paste
D	Fit to window		J	Layout
E	Restore to 100%		K	Up One Level
F	Plot Selected Data		L	Help
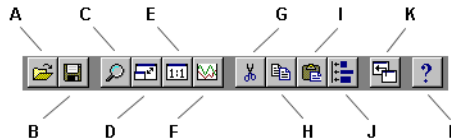
Zoom

Select **View > Zoom In** or click the Zoom button to change the mouse pointer to a crosshair. Use this pointer to drag an outline around the area you want to enlarge.

Restore to 100%

Select **View > Restore to 100%** or click the Restore to 100% button to return to the normal view.

Fit to Window

Select **View > Fit to Window** or click the Fit to Window button to force the user diagram to fit in the current window.

Up One Level

Select **File > Up One Level** or click the **Up One Level** button to return to the previously displayed window. Note that this also closes the current window. If you want to be prompted when you reach the top level of a diagram, make sure **Options > Browse closes active window** is selected.

Workspace

The main area in the center of the Vista screen is the workspace. The workspace is used to display information such as user diagrams and data and event viewers.

Status Bar

The status bar displays:

- The status line, which describes any active processes and provides a brief description of the currently selected command or toolbar button.

- The current time on the workstation.

- A progress indicator bar.

- Two communication status lights.

Toolbox

In Edit mode, the toolbox also appears on the screen. You can add objects to the user diagram using the toolbox. See Creating and customizing a user diagram for more information on the toolbox and Edit mode.

Other display features

Vistaoffers several display features to assist you when viewing a user diagram. Some of these are accessible through the toolbar and menu options (see Toolbar). Those only accessible through menu options are:

Browse closes active window

Select **Options > Browse closes active window** to have each window that you open replace the previous one. This option is selected by default. Without this option selected, any previously opened windows remain open.

Arrange All

Select **Window > Arrange All** to tile all open windows in the workspace.

Display mode and Edit mode

There are two display modes in Vista, which affect how you navigate a user diagram. Whether or not the toolbox is shown indicates the mode you are in: if the toolbox is open, you are in edit mode; if it is closed, you are in display mode.

- **Display** mode allows you to monitor your system, view system data, and control some aspects of your system. In Display mode, **single-click** objects to view the information they contain or perform their associated action.

- **Edit** mode allows you to configure the appearance and function of user diagrams and diagram objects. In Edit mode, **double-click** objects to view the information they contain or perform their associated action. **Right-click** the objects to view configuration options. See Creating and customizing a user diagram and Customizing a diagram object for more information.

When instructed to click a diagram object, either single-click or double-click, depending on the mode you are in.

Vista user diagrams

This section explains how to use Vista user diagrams to view real-time and logged data from your power monitoring system. The information that you obtain from your WinPM.Net network is represented graphically in a user diagram.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

What is a user diagram?

A user diagram is a configurable display that consists of two main elements:

- **Diagram objects** are configurable graphic icons that display your system information. Each diagram object displays a single piece of information from somewhere in your power monitoring network. In some cases, diagram objects are stored within grouping windows.

- **Grouping object**: In some cases, a user diagram stores related information within a grouping object. Click on a grouping object to open a grouping window and display its contents.

    NOTE: To configure a grouping object to open a grouping window, see Specifying an action for numeric, status, or grouping objects.

- **Grouping windows** are a type of folder or "sub-window" that contain diagram objects. Vista uses grouping windows to logically organize diagram objects within a user diagram. When closed, a grouping window is usually represented by a grouping object icon.

TIP: It is better to use grouping objects to open other user diagrams rather than to open grouping windows. Large numbers of grouping windows in a user diagram can slow the performance of Vista.

You can create as many user diagrams as you want and edit each one to suit your needs. You can share user diagrams with multiple users over your network, and users can view them simultaneously and share the same information. For more information on creating user diagrams, see Creating and customizing a user diagram.

Default user diagrams

Vista provides a set of pre-configured user diagrams that you can use to display data. These default diagrams provide a standard set of options that you can use to access your device's most commonly required functions.

The default diagram displays real-time data. It includes a simple power system illustration that displays the various real-time parameters measured by the device.

Generating, opening and closing a user diagram

A user diagram is identified by its ".dgm" extension. You can generate, open, and close user diagrams as needed.

Generating a network diagram

The network diagram displays your entire power monitoring system by linking to individual default user diagrams that display data from each device in your system. When you generate a network diagram, Vista automatically locates all devices in your system and displays them. This diagram is saved with the default name "network.dgm".

1. Select **File > Generate Network Diagram**.

   The network diagram appears in the workspace containing icons that represent groups of devices on your system.

2. Click on one of the grouping objects to open a window and display the node icons contained within.

3. Click a node icon to display the user diagram for the associated device.

4. Save the network diagram.

NOTE: Regenerating a network diagram and saving it overwrites the previously saved (network.dgm) file.

Opening a user diagram

To open a user diagram:

1. Select **File > Open**.

2. Navigate to the user diagram you want to open and select it from the **Open Diagram** dialog.

3. Click **Open** to open the selected user diagram.

Specifying the user diagram that opens for a given user

To set a specific Vista workspace and set of diagrams to open every time a given user logs on:

1. Log on to Vista using the particular user's logon credentials.

2. Organize Vista to appear as you want it to appear on startup.

3. Click **Save**, then close Vista.

4. Use Windows Explorer to navigate to the ...\WinPM.Net\config\cfg\ud folder.

5. Locate the .wsu file for the user you are setting up (i.e., <username>.wsu). Right-click and select **Properties**.

6. Select the **Read-only** attribute, then click **OK**.

> **NOTE**: Applying the Read-only attribute prevents the workspace information from being modified, so Vista opens the same way for a given user every time. The user can still modify and save user diagrams (if allowed by their access level).

Closing a User Diagram

To close a user diagram:

- Select **File > Close** to close the current diagram.

- Select **File > Close All** to close all diagrams.

Navigating a user diagram

After you open a user diagram, you can use it to interact with your power monitoring system. Each user diagram displays system information through diagram objects, which you can use to monitor and control various aspects of your system.

Monitoring your system in Vista

Use Vista user diagrams to display real-time and logged data, monitor events and alarms, and control a variety of system functions.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software to control time-critical functions.
- Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

How data is displayed

Information in your user diagram is displayed using diagram objects. Each diagram object communicates a single piece of information from somewhere in your power monitoring network; for example, a single numeric object displays numeric data from a single source.

There are seven types of diagram objects: grouping, text, numeric, status, data log viewer, event log viewer, and control. Each type of object performs a specific type of function.

In most cases, you can identify an object's function by a caption, a label, or both (the text object does not use a caption or a label).

- By default, the caption is usually located below the diagram object and identifies the node to which the diagram object is connected. The caption can be modified to display a custom description. See Caption options for more information.

- When a label is used, it is usually displayed to one side of the diagram object. By default, the label identifies the register (on the node) to which the diagram object is connected. The label can be modified to display a custom label name. See Text options for more information.

TIP: To toggle between default labels and custom labels, select or clear **Options > Show Default Labels**. You can identify the type of diagram object by pointing to it with the mouse. The object type appears on the status line.

Identifying stale data

Real-time data that has not been refreshed is referred to as stale data. Vista detects stale data as it occurs and identifies it with a colored border. By default, Vista identifies data as stale after 60 seconds. Position the mouse pointer over an object to display a ToolTip that explains why the data has become stale.

The stale data settings are user-configurable on a per-diagram basis, and can be customized by anyone with the necessary access level.

Changing the Stale Data settings

To change the amount of time (in seconds) that Vista uses to identify stale data:

1. Ensure that the Toolbox is open (**Options > Show Toolbox**), right-click the background of the diagram and click **Properties**.

2. Click the **Updates** tab and change the settings in the **Stale Data** section.

3. Click **OK**.

Changing the flag settings for Stale Data

To change the flag color:

1. Click **Options > Flag Colors**.

2. Click the **Edit** button for the flag color you want to change, select the new color and click **OK**.

Identifying errors in communication

Vistadetects errors (such as disabled devices, security access restrictions, or configuration errors) and identifies these with a colored border around an object (orange, by default). The border color can be customized (see the previous section, Changing the flag settings for Stale Data). Position the mouse pointer over an object that Vista is indicating has an error to display a ToolTip that explains the error.

Increasing the timeout period of a Vista query

When Vista queries the ION_Data database, the query can timeout if the database is large.

To increase the timeout period of a Vista query, a new entry to the registry of ION_SERVER needs to be created, and the Query Server connect timeout period needs to be increased.

NOTICE

IRREVERSIBLE OPERATING SYSTEM DAMAGE OR DATA CORRUPTION

Before making any changes, back up your Windows Registry in a network folder or other remote location.

Failure to follow these instructions can result in irreparable damage to the operating system of the computer and all existing data.

NOTE: Registry edits must be performed only by qualified and experienced personnel.

Increase the Vista query timeout

1. Select **Start > Run** and type regedit in the **Run** window to open the registry.

2. Navigate to HKEY_LOCAL_MACHINE\SOFTWARE\Siemens Industry\WinPM.Net\x.x, where x.x is the version number.

   If the component is run on a 64-bit operating system, navigate to:

   HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Siemens Industry\WinPM.Net\x.x

3. Right-click an empty area in the pane on the right and select **New > DWORD Value**.

4. Name the new DWORD VistaQueryTimeout_s.

5. Right-click VistaQueryTimeout_s and select **Modify**.

6. Enter a value for the timeout period.

   The timeout period is 60 (for 60 seconds) by default. You can increase the value to 120 initially. If timeouts still occur, modify the value to 180.

7. Select **File > Exit** to exit the registry editor.

NOTE: If your system installation includes Client machines running Vista, you need to update the registry for each one.

Increase the Query Server timeout

1. Open the WinPM.Net folder and double-click the Designer icon to start Designer.

2. Select **File > Open > QUERYSERVER.host_name** (where host_name is your workstation name) to open the Query Server node.

3. Double-click the **Query Modules** grouping object, then right-click the query module to open its setup dialog.

4. Select the Connection String setup register and click **Modify**.

5. Change **Connect_Timeout** from the default 60 to 120 (or 180 if necessary).

6. Save your change and restart the Query Server Service.

Viewing real-time data

Vista uses numeric objects and status objects to display real-time data. Numeric objects display real-time numeric values and status objects display real-time Boolean (ON/OFF) information.

Viewing data with numeric objects

Vista uses numeric objects to display real-time numeric data from your power monitoring system. Numeric objects can display many types of data, such as line voltage or current, power or energy levels, harmonics, or demand parameters.

Depending on their configuration, numeric objects can display data in different formats: alphanumeric, dial, horizontal or vertical bar, or scrolling graph.

See Customizing the display of a numeric object for more information on configuring numeric objects.

Using flags on a numeric object

In some cases, a numeric object may use flags to indicate the low and high limits of the displayed parameter.

When used on a dial display, flags appear as colored areas on the dial. When used on other types of numeric objects, flags change the object's background color to indicate that a Low, Low Low, High, or High High limit is currently exceeded. To view or change the default colors, select **Options > Flag Colors**.

The following image depicts a Low Limit of 140, a Low Low Limit of 60, a High Limit of 260, and a High High Limit of 340. The portion of the dial between 140 and 260 is transparent in Vista (the background color fills that portion of the dial), and is white in the Web-based Diagrams application.
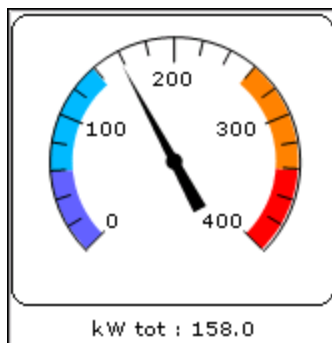
TIP: If you need only Low Low and High High limits, set the Low limit equal to the Low Low limit and the High limit equal to the High High limit.

Flag colors are stored in the workspace file on a per-user basis. They are configurable and can be customized by anyone who has the necessary access level. See Display options for more information on configuring flags.

Viewing data with status objects

Vista uses status objects to display Boolean (ON/OFF) conditions in your system. A status object may display the status of a digital output (such as a relay), a digital input (status input), or a setpoint in your system.

The default status object appears as an indicator light that changes color to indicate various conditions. Gray indicates the status object is not linked or not receiving data.

Like most diagram objects, status objects can be modified by any user with an appropriate access level. Depending on how a status object has been configured, its appearance and function can differ significantly from the default settings.

You can specify that it appear as an alphanumeric display or you can select custom images representing different conditions, for example for ON, OFF, and Unconnected. See Customizing the display of a status object for more information.

> **NOTE**: In some cases, the active state of a status object may have been changed (inverted) so that an active state appears as inactive and vice versa. See Customizing a diagram object for more information.

Viewing logged data

Data and events recorded by the devices in your system are regularly transferred by the Log Inserter from each device to the database. Vista uses the WinPM.Net Query Service to transfer this information from the database to your workstation. Vista provides two ways to view logged data and events: the Data Log Viewer and the Event Log Viewer.

- The Data Log Viewer displays data logs from all devices represented in the active user diagram. You can use the Data Log Viewer to display waveform and trend data in tabular format. You can also select a range of logs from the viewer and display the data graphically in the log view plotter.

- The Event Log Viewer typically displays events and alarms from each device represented in the active user diagram. The Event Log Viewer is used primarily to monitor low-priority events and alarms associated with the active user diagram. Each Event Log Viewer is associated with a single user diagram.

Using a Data Log Viewer

Use a Data Log Viewer to view system data stored in the database. Data Log Viewers display historical trend data (from data recorders) and waveform data (from waveform recorders).

You can select any data displayed in a Data Log Viewer and plot it as a graph or copy it to another application (for example, Microsoft Excel) where you can perform advanced calculations or include it in a report.

See Using the Query options for information on log viewer configuration.

Displaying a Data Log Viewer

Click on a Data Log Viewer object in your user diagram. The Data Log Viewer appears.

Each Data Log Viewer displays data as a table of columns and rows. Each labeled column contains a single type of data and each numbered row represents a single data record.

Plotting a range of data from the Data Log Viewer

You can plot data from a Data Log Viewer in the Log View Plotter.

1. Highlight the cells containing the data that you want to plot. The range of data can be selected in one of three ways:

   - To graph trends for all parameters over a particular time span, select a group of rows.

   - To graph the trend for a specific parameter using all available records, select the entire column.

   - To graph multiple parameters over a specific time span, click on the first cell in the desired range then drag to highlight the last cell of the range.

2. Select **Edit > Plot Selected Data** or click ⊡.

   The Log View Plotter window appears, displaying a graphical representation of the selected data. See Plotting logged data for more information.

> **TIP**: To quickly plot a single waveform, double-click on its waveform symbol in the Data Log Viewer.

Copying a selection to another application

1. Highlight the cells containing the data that you want to copy.

2. Select **Edit > Copy** or click ⊡. This copies the selection to the Windows Clipboard.

3. Switch to another application and paste the selection.

Using the Event Log Viewer

Events and alarms that occur in your system are recorded by the meter and transferred via Log Inserter to the database.

Vista. Use the Event Log Viewer to view event records. The Event Log Viewer displays events and alarms from the device(s) represented in the active user diagram. Use it to monitor low-priority events and alarms associated with the active user diagram. The Event Log Viewer is associated with a single user diagram, so alarms only update when the user diagram is open.

See Using the Query options for information on log viewer configuration.

Displaying an Event Log Viewer

⊡ Click an Event Log Viewer object in your user diagram to open the Event Log Viewer.

The Event Log Viewer displays logged events in a table of columns and rows. Each labeled column contains a single category or event description and each numbered row represents a single event record.

Typically, Event Log Viewers display the following information:

- **Timestamp**: This indicates the event's date and time.

- **Priority**: This determines the relative significance of the event. Each type of event has a numeric value assigned to it that represents its relative priority level.

- **Cause**: This is the label and value describing the cause of the event. For example, if the label is "Setpoint #1" and the value is "Active," then the event was caused by setpoint #1 changing to an active state.

- **Effect**: This is the label and value describing the effect of the event. For example, if label is "Relay #1" and the value is "Forced On," then the effect of the event was to force relay #1 on.

- **Ack Time**: This is the date and time that an alarm was acknowledged.

> **NOTE**: Because Event Log Viewers are associated with a single user diagram, they only update when the user diagram is open. All Event Log Viewers need to be re-opened if a diagram is closed or the software is shut down.

Acknowledging alarms

Every type of event has a prioritized value that identifies its relative significance on a scale of 0 (zero) to 255. By default, Vista identifies any event with a priority of 128 (default) or greater as an alarm.

The event record is highlighted in red, and identified under the **Ack Time** column as "ALARM". The **Cause** column lists the source of the alarm and the **Effect** column describes the event.

By default, Vista plays a "beep" to annunciate an event with a priority range of 128-191, and a beep combined with a flashing display to annunciate an event with a priority range of 192-255. To customize these options, see Alarming options.

When you acknowledge an alarm, Vista records the time the alarm was acknowledged (in the **Ack Time** column) and your user ID (in the **User Name** column).

The alarm acknowledgement function is also available in the Alarms application in the Web Applications component. See the online help for the Alarms application for information about acknowledging alarms.

> **NOTE**: In some cases, your access level may not be sufficient to acknowledge an alarm; for example, a critical alarm may require Supervisor access. This is a user-configurable option. See Alarming options for more information.

Acknowledging an alarm

1. Open an Event Log Viewer, then do one of the following:
   - To acknowledge a single alarm, click the word ALARM in the **Ack Time** column or highlight the row containing the alarm.
   - To acknowledge all alarms to date, click the **Ack Time** column heading.

   A confirmation box appears, asking if you want to acknowledge the selected alarms.

2. Select **Yes** to acknowledge the alarms or **No** to cancel.

If you acknowledge the alarm but do not have a sufficient access level, the alarm remains unacknowledged and the message "Insufficient Authorization" appears in the status line.

NOTE: When an alarm has been acknowledged in the Web-based Alarms application, and you view the entry in the Event Log Viewer, the **Ack Time** and **User Name** columns are populated with the appropriate acknowledgement entries for the ON value. However, these columns remain blank for the related OFF value even though the alarm has been acknowledged.

Silencing alarms

If you are not ready or able to acknowledge an alarm, you can silence it until it can be acknowledged by selecting **Options > Silence Outstanding Alarms**.

NOTE: Acknowledging an alarm does not change the operating condition that caused the alarm. You must ensure that the cause is corrected to remove the alarm condition.

Viewing user diagram of devices with security

You can view the user diagram of devices with security settings enabled in Vista.

To view user diagram in Vista for devices with standard security enabled,

1. Select **File > Generate Network Diagram**.

   The network diagram appears in the workspace containing icons that represent groups of devices on your system.

2. Click on one of the grouping objects to open a window and display the node icons contained within.

3. Click a node icon to display the user diagram for the associated device.

NOTE: To perform any control object function, enter the device password. The user name is set to default. Refer for [Controlling system functions](#) more information. If user name is prompted when performing any control object function, then refer to [Changes in security settings of device](#) for more information.

To view user diagram in Vista for devices with advanced security enabled,

1. Select **File > Generate Network Diagram**.

   The network diagram appears in the workspace containing icons that represent groups of devices on your system.

2. Click on one of the grouping objects to open a window and display the node icons contained within.

3. Click a node icon to display the user diagram for the associated device.

4. Enter the user name and password of the device.

NOTE: All control object functions can be performed without entering credentials.

Controlling system functions

You can use control objects to send commands from Vista to a node.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

• Do not use the software to control time-critical functions.

• Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

Each control object in your user diagram is configured to perform a single function. When you click a control object, it performs the specified function. For example, you could click a control object to clear an energy accumulator, toggle a relay, reset a counter, or adjust the value of an analog output device.

The default appearance of a control object depends on the function it has been configured to perform:

| Boolean | Pulse | Numeric |

Regardless of their appearance, all control objects are used the same way—to use a control object, click it.

> **NOTE**: For information on configuring the message, appearance and access level of control objects, see Customizing the action for a control object and Customizing the display of a control object.

When you click on a control object, one of the following happens, depending on how the control object is configured:

• Vista performs the specified action immediately.

• Vista displays a confirmation message. Click **Yes** to proceed or **No** to cancel the operation.

• Vista requests a password. Type the logon password then click **OK** to proceed.

Depending on the function, a dialog prompting for the meter password may also open. If so, type the meter password then click **OK** to proceed.

> **NOTE**: There can be a time delay until Vista displays a new control value or state after it has been written to the device.

Plotting logged data

This section describes how to use and customize the Log View Plotter to view logged data in a graphical representation. Use the Log View Plotter to graphically show trends in numeric data, display logged waveforms, display phasor diagrams, and perform harmonics analysis on

waveform data. You can also plot Sag/Swell voltage disturbances and compare them to industry standard voltage tolerance curves (such as the CBEMA/ITIC curve).

> **TIP**: You can copy information displayed by Vista Log Viewers to the Windows Clipboard and paste it into other Windows applications.

Plotting logged data

1. Select data from a data log viewer:
   - To graph trends for all parameters over a particular time span, select a group of rows.
   - To graph a trend for a specific parameter using all available records, select an entire column.
   - To graph multiple parameters over a specific time span, select the first cell in the desired range and drag to highlight to the last cell of the range.

2. Select **Edit > Plot Selected Data** or click .

   Vistaplots the data and displays it in the Log View Plotter window.

The Log View Plotter interface

The Log View Plotter is a Vista window that graphically displays selected data.

The default display is the Plot Display tab. Depending on the type of data selected, the Log View Plotter may also offer a Harmonics Analysis tab and a Phasor Diagram tab.

In all cases, the Log View Plotter offers additional information in two floating boxes: the Legend and the Calculations window.

The Legend

Each line or bar in the graph is represented by a pattern of the same color in the legend.

> **TIP**: Click on the curve sample in the legend to select its equivalent in the graph.

Displaying the Legend

1. Right-click the background of the Log View Plotter window.
2. Select **Graph Options** in the menu to open the **Graph Options** dialog.
3. Select the **Show legend** check box then click **OK**.

The Calculations window

The Log View Plotter calculates several values for every line or bar in the graph. You can view the results of these calculations in the Calculations window.

The Calculations window updates automatically, displaying a range of calculated values based on the currently selected parameter. By default, all values displayed in the Calculations window are calculated from the first parameter. To view the calculations for a different parameter, select the line or bar you want.

For example, to view the calculations for a curve on the Plot Display tab, click on the curve line in the graph or on its line pattern in the legend. When you click a different tab, the Calculations window displays the values relevant to that tab.

Displaying the Calculations Window

1. Right-click the background of the Log View Plotter window.

2. Select **Graph Options** from the menu to open the **Graph Options** dialog.

3. Select the Show calculations check box then click **OK**.

Changing graph options

When using the Log View Plotter, you can adjust the graph options to change the title of a graph or the way data is displayed.

1. Right-click the graph background then select Graph Options. The **Graph Options** box appears.

   - Type a name for the graph in the **Title** box. This name appears in the title bar of the Log View Plotter window. The default is Log View Plotter.

   - Select or clear the **Show legend** and **Show calculations** check boxes to show or hide the legend and Calculations window.

   - Select the **Align triggers** check box to align the trigger times of all waveforms in the graph. (When multiple waveforms are plotted, their trigger times may not always be correlated.)

   - Select **Separate curves vertically** to display each curve separately.

   > **NOTE**: See Calculating Harmonics using more than one cycle for more information on the System Frequency and # of cycles for harmonics options.

2. Click **OK** to save your changes.

Displaying curve data in a table

After you plot data in the Log View Plotter, you can select a curve and display its data points in a table. This option is only available on the **Plot Display** and **Harmonic Analysis** tabs.

1. Right-click the curve line in the **Plot Display** or **Harmonic Analysis** tab and select **Data** from the menu.

   - **Plot Display** tab: The data points are displayed. The X column lists the x-axis coordinates and the Y column lists the y-axis coordinates. The first column (#) assigns a number to each set of points.

   - **Harmonic Analysis** tab: A window appears listing each harmonic number and the magnitude of each harmonic for each parameter. The first column numbers the rows. The X column lists the harmonics number and the Y1, Y2, Y3, etc., columns list the harmonic values for each bar in the harmonics histogram. Rows where the X value is not an integer (i.e., .5, 1.5, 2.25) contain sub-harmonic values.

2. Click **Format** to change the number of significant digits displayed in the X and Y columns or to change the width of these columns.

   - Width defines the number of characters the column can display.

   - Precision defines the number of significant digits displayed.

3. Click **Copy** to copy columns to the clipboard. This allows you paste a copy of the data into another application, such as a spreadsheet program or a text file.

> **NOTE**: The Log View Plotter can only display one data table at a time. If you want to display tabular data for another curve (from the same graph or a different one), you must first close any open data window.

Viewing the Plot Display tab

The **Plot Display** tab depicts the specified parameters as curves against a graph.

Each parameter is plotted in a different color. The legend identifies the color of each curve in the selection and what it represents. Click on a curve to select it or right-click it to display additional options. The timestamp at the top of the tab identifies the first point in the selected curve.

The vertical axis (y-axis) represents the specified range; the horizontal axis (x-axis) represents time. Vista automatically adjusts the scale of the y-axis and x-axis to accommodate the largest parameter(s) in the selection. Right-click either axis to change the scale of its properties; right-click the axis numbers to change their format.

The time unit indicator in the lower right corner identifies the time scale of the x-axis (for example, "h" indicates an hourly scale). Possible time scale units include months, days, hours, minutes, and seconds.

Zooming in on the Plot Display tab

To zoom in on the **Plot Display** tab, click on the Zoom button 🔍 then drag a selection box around the area you want to view. To view the entire plot, click the Restore to 100% button ⬚.

Calculations window parameters for the Plot Display tab

The Calculations window updates automatically, displaying the following values for the currently selected curve:

Value Label	Description
Cursor 1	The timestamp and the y-value of the selected curve at Cursor 1.
Cursor 2	The timestamp and the y-value of the selected curve at Cursor 2.
Delta C	The time difference between Cursor 1 and Cursor 2, and the difference between the y-values at Cursor 1 and at Cursor 2.
Min	The minimum value of the selected curve between the two cursors.
Max	The maximum value of the selected curve between the two cursors.
dpeak	The maximum peak-to-peak value of the selected curve between the two cursors (Max-Min).
Avg[1]	The average value of the selected curve between the two cursors.
RMS[1]	The root-mean-square value of the selected curve between the two cursors.

Value Label	Description
Int(H)[1]	The integration of the selected curve between the two cursors with respect to time in hours (for example, if the curve is in kW, the information is displayed in kWh.)

[1] The last three values (Avg, RMS, INT(H)) on the Calculations window are only available on plots where the x-axis represents time. They are not offered on other types of plots (for example, CBEMA plots).

Both numeric data and waveform data can be plotted in the same graph. On waveform plots, the trigger time is represented by a red vertical line.

Using the cursor lines

There are vertical cursor lines at either end of the curves in the Log View Plotter. These cursor lines define the start and end points used by the Calculations window. By default, Cursor 1 is located on the leftmost point of the selected curve and Cursor 2 on the rightmost point.
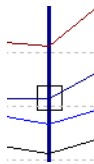
To define a new start point, drag Cursor 1 along the curve to the new start point. Repeat this procedure with Cursor 2 to define a new end point. You can also use the arrow keys to move the active cursor left or right. The active cursor is identified by a small box that appears where the cursor line meets the curve. Use the TAB key to switch the active cursor between Cursor 1 and Cursor 2.

As the cursor line moves from point to point along the curve, the information in the Calculations window is updated. You cannot drag Cursor 1 past Cursor 2 or vice versa.

De-indexing parameters and specifying Y-offsets

If you are plotting multiple parameters with different magnitudes, the lower magnitude values may be difficult to see in the default graph. Vista allows you to index any parameter to a different axis that can be scaled and offset independently from the other axes.

The new axis automatically assumes a scale appropriate for the selected parameter. In this way, you can plot parameters of different magnitudes on the same graph (such as voltage and current).

De-indexing a parameter or offsetting a parameter on the Y-axis

1. Right-click the graph background to display the menu. In the last section of the menu, highlight the curve you want to change.

   The curve's sub-menu appears. (You can also access the sub-menu directly by right-clicking on the parameter curve or on its curve sample in the legend.)

2. Select **Options** from the sub-menu to open the **Curve Option** dialog.

3. Choose one of the four axis options for the selected parameter. The axis is indicated by the location of the tick marks.

Specify a Y-offset if required. This shifts the selected parameter up the y-axis (or down for a negative number) by the amount you specify.

4. Click **OK**. A new axis appears with a default title and a scale appropriate for the selected parameter.

   The scale of the default axis can also be adjusted to accommodate the remaining curves. A parameter has an asterisk after it in the legend if it has been offset.

Repeat these steps for each parameter that you want to de-index from the default axis. You can assign more than one parameter to an axis and the scale adjusts accordingly to incorporate each new parameter. For example, if you have plotted three voltage parameters and three current parameters, you can select the current curves and assign each of them to another common axis, leaving the three voltage curves to share the default axis.

Displaying Power Factor data

If you have plotted power factor data (from historical logs) in the Log View Plotter, you can improve the display to avoid discontinuity by setting the vertical axis to Power Factor Display (-100,+100).

Setting any vertical axis for Power Factor display

1. Right-click the vertical axis or right-click the graph background and select the axis after you have plotted the power factor data.

2. Select Power Factor Display from the menu. A check mark appears beside the option to indicate that it is selected.

You can plot other non-power factor data on the same graph; however, you should not plot the new data on the same axis as the power factor data. De-index the new data on a separate axis as described in <u>De-indexing a parameter or offsetting a parameter on the Y-axis</u>.

Viewing the Harmonics Analysis tab

If any of the parameters you have plotted is a waveform, you can display a histogram of the signal's harmonic content by selecting the Harmonics tab on the Log View Plotter window. The resulting histogram shows the harmonics for all waveforms in the plot.

> **NOTE**: The cycle used is the first full cycle to the right of Cursor 1on the Plot Display tab.

The total harmonic distortion (THD), K-factor and Crest factor values for this waveform are displayed in the Calculations window.

To view the calculations for another parameter, click on it in the graph or on its curve sample in the legend. The Calculations window updates automatically. The currently selected parameter is indicated above the calculations.

Calculating Harmonics using more than one cycle

By default, the harmonics are calculated using one cycle. You can increase the number of cycles by any power of two cycles (i.e., 2, 4, 8, 16, 32 or 64) up to the maximum number of cycles available for the waveform.

1. Right-click the background of the graph area then select **Graph Options** to open the **Graph Options** dialog.

2. Select the number of cycles over which harmonics calculations are to be performed from the "# of cycles for harmonics" list.

3. Select the appropriate frequency in the "System Frequency" box, if the waveform you are analyzing came from a system with a frequency other than 50Hz or 60Hz. (For example, if the waveform was captured from a 400 Hz system using a 3710 ACM, specify "400Hz" in this box.)

> **NOTE**: The Harmonics Analysis tab is only available for waveform records. Vista does not perform a harmonics analysis for trend data or for waveforms with less than eight samples per cycle. You cannot display the harmonics of more than 16 parameters at a time.

Viewing the Phasor Diagram tab

If you have plotted waveforms for all three voltages and/or currents of your system, you can view this information as a phasor diagram. The phasor diagram graphically displays the three phases (voltages and/or currents), showing their relative magnitudes and angles.

You can use a phasor diagram to evaluate important aspects of your power system such as voltage balance, per-phase loading, and type of load (or generator operating mode). A phasor diagram also provides a way to identify PT or CT wiring problems.

> **NOTE**: The timestamp at the top of the Phasor Diagram tab is based on the time of Cursor 1 on the Plot Display tab.

To display a phasor diagram, select the Phasor Diagram tab from the Log View Plotter window.

The Phasor Diagram and calculations are derived from a one-cycle window starting at Cursor 1 on the Plot Display tab. The magnitude and angle of the selected phase is displayed in the Calculations window.

> **NOTE**: The Log View Plotter conforms to the convention where all phasors are plotted with respect to V1 (always at 0°) and rotate in a positive counter-clockwise direction. On a balanced power system, all three phases should appear 120 degrees apart.

To edit a phasor displayed on a phasor diagram, right-click a parameter in the legend to display a menu where you can change the phasor's color or delete the phasor from the diagram.

Symmetrical components

The Log View Plotter calculates the symmetrical components of your power system — the positive, negative, and zero sequences relative to the first phase — for the cycle selected. If you have plotted waveforms for three voltages and/or currents, you can access their symmetrical component calculations. To produce correct symmetrical components, the waveforms must have been plotted in the correct order (for example, V1, V2, V3 — not V1, V3, V2).

The analysis of an unbalanced system, such as determining the effects of a system fault, is made simpler by using symmetrical components. Symmetrical components are a mathematical tool that allows any system of three unbalanced phasors to be represented by three balanced phasor systems. The total current or voltage in any phase wire is expressed as the sum of three balanced, three-phase components.

Positive Sequence Components (1) - these consist of three phasors that are equal in magnitude and displaced from each other by 120º and have the same phase sequence as the original phasors.

Negative Sequence Components (2) - these consist of three phasors that are equal in magnitude and displaced from each other by 120º and have the phase sequence opposite to that of the original phasors.

Zero Sequence Components (0) - these consist of three phasors that are equal in magnitude and with zero phase displacement from each other.

If the original phasors of voltages are Va, Vb, and Vc, then the symmetrical components would be as follows:

- Va = Va1 + Va2 + Va0

- Vb = Vb1 + Vb2 + Vb0

- Vc = Vc1 + Vc2 + Vc0

The symmetrical components function, as implemented inside some ACCESS meters, is useful for steady state analysis (unbalance), since the parameters are updated every second. However, when determining the symmetrical components during a fault, per-cycle analysis is required. For this application, you do not need real-time updates from the meter, because you are only interested in the relatively short time before, during, and after a system fault. This information is contained in the waveforms recorded when triggered by a disturbance.

Viewing symmetrical components

1. From a waveform Data Log Viewer (such as Waveforms/Sequence of Events), select all three phases of a fault (voltage or current).

2. Select **Edit > Plot Selected Data**.

3. Drag the Cursor and move it to the location of the fault.

4. Click on the **Phasor Diagram** tab to view the Symmetrical Components in the Calculations window.

Plotting waveforms from devices with high sampling rates

Waveforms from devices with high sampling rates can be quite large. These large waveforms are split into a series of smaller waveforms when displayed in the data log. To plot all the sections of the waveform in Vista, you must select all the rows that relate to the waveform before you click the Plot button. All the rows related to the one waveform have the same timestamp.

When the waveform is displayed, you may need to zoom in to view the area of the waveform you are interested in. To zoom in, drag a selection box around the area you are interested in.

Adding parameters to a Log View Plotter

After you have created a graph and the Log View Plotter window is visible, you can add additional parameters to the graph by copying and pasting them into the graph. These parameters can be copied from the same data log viewer, a different data log viewer, or from a different Windows application such as Microsoft Excel. The x and y-axes are automatically re-scaled to accommodate the new parameters.

When adding waveforms to the log view plotter, you can correlate all of their trigger times. See [Overlaying curves in the Log View Plotter](#) for more information.

Adding data from a Data Log Viewer

You can add additional parameters from any data log viewer configured to provide compatible time-based data. Any added parameter is plotted using the same x-axis used by the existing curves. To avoid confusion ensure that the column you selected for your x-axis in the data log viewer (from which you are adding data) contains the same parameter (typically a timestamp) as the original data log viewer.

Adding additional parameters from a Data Log Viewer

1. Select the range of cells you want to plot. Ensure that it contains the same type of information (timestamps) as the data already being plotted.

2. Select **Edit > Copy**.

3. Click on the title bar of the existing Log View Plotter window then select **Edit > Paste**. The parameters you selected are added to the graph.

If you want to plot waveforms on the same graph as historical data, the historical data should originate from high-speed data recorders and span a short time range, typically a few waveform cycles (<1 second).

Adding events from an Event Log Viewer

You can add events to a waveform or trend plot. This is useful for comparing abnormalities of the waveform or trend shape with any corresponding events that happened in the same time period.

1. Select the row(s) you want to add in the Event Log Viewer.

2. Select **Edit > Copy**.

3. Click on the title bar of the existing Log View Plotter window then select **Edit > Paste**. The copied event records are added to the graph.

Each event is represented in the Log View Plotter as a diamond symbol. A diamond symbol is added to the legend and diamond symbols appear at the appropriate areas along the top of the diagram.

Point to a diamond symbol to display a description of the event in the status bar at the bottom of the screen.

Adding data from a spreadsheet or text file

You can select data from a spreadsheet or text file and paste it into an existing graph. This is useful if you want to compare trends from different times, waveforms showing different system events, and other analysis applications.

Ensure that the first column of data in the spreadsheet or text file is the same type as the first column in the data log viewer from which the log plot originated. In particular, for timestamps, make sure the entire timestamp is contained in a single column and that it uses the same date and time format as defined in the Windows Control Panel. If the date portion of the timestamp is in one column and the time is in another, or if the format is different in any way, the data cannot be pasted into the Log View Plotter.

When you paste data into a graph, from both spreadsheets and text files, the Log View Plotter interprets the first row of data in the selection as a heading. The first row in your selection is used to label the parameter in the legend. So, if the first row is not already used for column headings, insert a row and type in the label you want to appear in the Log View Plotter.

To copy data from a text file, the text file must be formatted correctly. Each column in the text file must be separated by a tab and there must be a hard return at the end of each row.

Adding data to a graph from a spreadsheet or text file

1. Open the spreadsheet or text file.

2. Copy the range of values you want to plot, including the first column and the first row. Include other columns in your selection, if required.

3. Click on the title bar of the Log View Plotter window then select **Edit > Paste**.

    The parameters you copied and pasted are added to the graph. The axis scales are adjusted automatically to include the new data.

You can define a trigger for waveform records containing timestamps when you copy records from other applications. Simply add an exclamation mark to the end of the timestamp to indicate that it is the trigger time. For example, the exclamation mark at the end of 22/05/2015@02:00:41.035 PM! indicates that the trigger occurred at this time.

Customizing the Log View Plotter display

Many display attributes for each curve can be adjusted. In most cases, if you right-click an area of the graph, either a menu or a dialog opens with configuration options. Unless otherwise noted, the options you specify for the Log View Plotter window are saved and restored every time you plot data from the original data log viewer. If you plot data from a different data log viewer, its graph has its own optional settings.

Customizing an axis

After you have plotted data in the Log View Plotter, use the following instructions customize the axis or gridlines, change an axis label's format or add an axis title, or change the font or color used on all axes. You can also change the scale of an axis.

Configuring an axis

1. Right-click the graph background to display the menu. Highlight the axis from the menu to reveal the sub-menu. You can also right-click the axis to access the sub-menu directly.

2. Select **Axis Properties** from the sub-menu to open a dialog with the following options.

- **Range**: Make changes in the **From** and **To** boxes to adjust the axis range.

- **Ticks**: Make changes in the **Major every** and **Sub-divided by** boxes to adjust the frequency of major and minor ticks respectively.

- **Grid**: Select or clear the **Major** and **Minor** check boxes to turn graph gridlines on or off. Click the **Style** buttons to change to the line style or width of major and minor gridlines.

- **Logarithmic scale**: Select this check box to enable the logarithmic scale option.

- **Line**: Click this button to display the **Line Parameters** dialog and change the line style or the width of the axis.

3. Click **OK** to save your changes.

> **NOTE**: The axis range and the frequency of the major ticks are not stored in the diagram but are recalculated for every new plot.

Changing the color of an axis or the gridlines

1. Right-click the graph background to display the menu. Highlight the axis from the menu to reveal the sub-menu. You can also right-click the axis to access the sub-menu directly.

2. Select **Axis Color**, **Major Grid Color**, or **Minor Grid Color** from the sub-menu to open the **Color** dialog.

3. Select the color you want or click **Define Custom Colors** for more options.

4. Click **OK** to save your changes.

   The axis or the major or minor gridlines change to the color you selected.

Changing the format of the axis labels or adding an axis title

1. Right-click the graph background to display the menu. Highlight the axis from the menu to reveal the sub-menu. You can also right-click the axis labels to access the sub-menu directly.

2. Select **Axis Labels** from the sub-menu to open the **Axis Labels** dialog.

3. Select the format in which you want the axis labels to appear.

   Decimal is used by default (for example, .01, .02, .03) but if the labels are too large or too small, select Scientific (1.0 x 10-2, 2.0 x 10-2, 3.0 x 10-2...) or Engineering (1m, 2m, 3m...).

4. Determine the axis title: select None to have no axis title or Text to enter your own title.

   For time-based plots, the default x-axis title is "s" if the timestamps of the data spans a number of seconds, "mn" for minutes, "h" for hours, "d" for days, "m" for months. (This applies to the Plot Display tab only.)

   For non time-based plots (CBEMA or harmonics), you can change the title. Select Text and type the title in the edit box. The title must be nine characters or less.

5. Click **OK** to save your changes.

> **NOTE**: If the first column of the source data includes a space or @ sign, the Log View Plotter assumes time on the horizontal axis and automatically determines the units based on timestamp information. The axis title is updated whenever the window is updated.

Changing the font used on all axes

1. Right-click anywhere in the graph background and select **Graph Font** to open the **Font** dialog.

2. Specify the font, style, size, effects and color options that you want to use for the axes labels.

3. Click **OK** to save your changes.

Customizing curve style

When displaying multiple parameters, Vista distinguishes between parameters using color. Each parameter is listed in the graph legend accompanied by a sample of its associated curve. The color of a curve can be changed after you have plotted data in the Log View Plotter.

If color is not enough to distinguish one curve from another—for example, if you plan to print the graph on a black and white printer—you can change the curve style after you have plotted data in the Log View Plotter.

By default:

- Each curve is shown by a thin line that connects each point of data. You can customize the style of each curve independently and show or hide the coordinates of each data point.

- Harmonics are displayed in a bar graph. You can distinguish between parameters by customizing the display style of each parameter. Display options include bar type, hatching, width and color.

Use the following instructions to customize curve styles, bar styles, and curve colors.

Changing the style of a curve on the Plot Display tab

1. Right-click the graph background to display the menu. Highlight the curve from the menu to reveal the sub-menu. You can also right-click the parameter curve in the diagram or its entry in the legend to access the sub-menu directly.

2. Select **Curve Properties** from the sub-menu to open the **Curve Style** dialog.

    - Select the type of curve you want from the Type list. Vista offers seven curve types: a simple line, a line with symbols, a scatter plot of markers, horizontal or vertical bars, or 3-D horizontal or vertical bars.

    - Select **Spline** (available for Line or Line + Symbol types) if you want the line that connects each point to be a smooth curve. If you leave it cleared, each pair of points is connected by a straight line.

    - Select **Fill Area** (available for Line type) to fill the area under the curve with the line color. Clear the check box to leave the area under the curve transparent.

    - The **Marker** section is available for Line + Symbol and Scattered types of lines. Specify the shape, size, and style of the marker used for the points. The Style options apply only to geometric shapes (for example, circles, boxes, diamonds, etc.).

    - For any line type, click **Line** to change the width and the style (for example, solid, dotted, dashed, etc.) of the line. For bar graph types, a dialog opens where you can specify the hatching pattern and the width of the bars.

3. Click **OK** to save your changes.

Changing the bar style for the Harmonics Analysis tab

1. Right-click the bar (or on its curve sample in the legend) and select **Curve Properties** to open the **Bar Graph Configuration** dialog.

2. Select the type of bar to use from the Type list. This selection is applied to all bars in the graph. Three bar styles are available:

   – **Group Bars**: Shows bars side-by-side

   – **Deep Bars**: Shows bars one behind the other

   – **Stacked Bars**: Shows bars stacked one on top of the other vertically

3. Select a hatching style for the selected bar from the Hatching list. This selection is applied to the currently selected bar only.

4. Set how wide you want each bar to be in the Width text box. The specified width is applied to all bars in the graph. The default depends upon the number of parameters.

5. Click **OK** to save your changes.

Changing the color of a curve

1. Right-click the graph background to display the menu. Highlight the curve from the menu to reveal the sub-menu. You can also right-click the curve in the diagram or its entry in the legend to access the sub-menu directly.

2. Select **Curve Color** from the sub-menu to open the **Color** dialog.

3. Select the color you want or click **Define Custom Colors** for more options.

4. Click **OK**. The curve changes to the color you selected.

Customizing graph and legend options

After you have plotted data in the Log View Plotter, you can change the color of the graphing area (and the legend) and define a font for the legend.

Changing the color of the graphing area

1. Right-click anywhere in the graph background and select **Background Color** to open the **Color** dialog.

2. Select the color you want or click **Define Custom Colors** for more options.

3. Click **OK**. The graphing area changes to the specified color.

Changing the font used in the legend

1. Right-click the background of the legend to open the **Font** dialog.

2. Specify the font, style, size, effects, and color options that you want to use for the legend text.

3. Click **OK** to save your changes.

Removing graph parameters

After you have plotted data in the Log View Plotter, you can remove a parameter from the graph.

1. Right-click the graph background to display the menu. Highlight the curve you want to remove from the menu to reveal the sub-menu. You can also right-click the curve in the diagram or its entry in the legend to access the sub-menu directly by right-clicking on the curve.

2. Select **Delete Curve** from the sub-menu. The curve is deleted without further prompting.

Overlaying curves in the Log View Plotter

After you plot data in the Log View Plotter, you can overlay a curve onto your graph to help analyze sag/swell data. This applies to devices that measure disturbances in voltage and store the magnitude and duration of these disturbances. When these values are inserted into the ION_ Data database, they provide a series of coordinates that can be plotted on a voltage tolerance curve. A voltage tolerance curve is often used to categorize the severity of a voltage disturbance.

NOTE: Most Vista queries are plotted against timestamps (with the timestamp as the x-axis); however, any query that uses the ION Sag/Swell module must be plotted against duration. When plotting sag/swell data, select Duration as the X-parameter of the Log View Plotter.

Vista provides common curves or voltage tolerance envelopes that can be used as overlays in the Log View Plotter:

- **CBEMA/ITIC**: These curves (for CBEMA, described by the IEEE standard 446-1987, and for ITIC, described by the IEEE 1100-1999 standard) define an upper and a lower bound on the types of disturbances that electrical equipment is likely to tolerate. It plots disturbance magnitude against disturbance duration and indicates at what points a voltage disturbance is likely to disrupt or damage equipment. Points near or outside the voltage tolerance envelope indicate events that could interrupt or damage electrical equipment.

- **SEMI F47-0200**: This specification defines the voltage sag tolerance capability required for semiconductor processing equipment, though it is now used by other industries. According to the specification, equipment must tolerate sags to 50% of nominal voltage for up to 200 milliseconds, sags to 70% of nominal voltage for up to 0.5 seconds, and sags to 80% of nominal voltage for up to 1 second. Points above the voltage tolerance curve indicate disturbances that exceed these specifications.

Overlaying curves on plotted data

To overlay one of the provided curves, you need a data log viewer that contains the magnitude and duration values for the sag/swell disturbances. The Duration column must be set as the X-parameter for the Log View Plotter—it appears in blue in the data log viewer table (See X-parameter for Log View Plotter for more information on setting the X-parameter for Data Log Viewers). The Magnitude values can be in any one of the other columns.

1. In the Data Log Viewer, select a range of cells to be plotted.

2. Select **Edit > Plot Selected Data**. The Log View Plotter window appears with Duration as the x-axis and Magnitude as the y-axis.

3. Right-click the graph and select **Overlay Curve** from the menu to open the **Overlay Curve** dialog.

4. Select the text file containing the curve data that you want to overlay (from the ...\WinPM.Net\config\lvp directory).

5. Click **Open**. The selected curve appears in the graph.

    You can review each point in your data and see where it falls in relation to the curve.

NOTE: This process automatically changes your x-axis scale to logarithmic.

Creating and customizing a user diagram

The following sections describe:

- How to customize an existing user diagram by customizing the appearance and function of a diagram object or diagram window.

- How to create a new user diagram.

- The various types of diagram objects and how to use each type to add function to your user diagram.

NOTE: You must have the appropriate security clearance to edit or create a user diagram. To determine if you can edit a diagram, select **Options > Show Toolbox**. If you can select this option, then you can edit a diagram. Consult your network administrator if you need to change your access privileges.

NOTE: Any changes you make to a default diagram affect all other devices using the same diagram.

Creating a new user diagram

To create a new user diagram, login to Vista and select **File > New**. A new (blank) user diagram appears. You can only create a new user diagram if you have supervisor-level access.

To automatically generate a network diagram based on devices added using Management Console, see Generating a network diagram.

Saving a user diagram

To save changes you made to a user diagram:

1. Select **File > Save** to save a new or existing user diagram, or **File > Save As** to open the **Save New Diagram** dialog.

2. Type a name for the diagram in the **File name** field and click **Save**.

By default, Vista points to the root of the \ud directory when you attempt to save a new diagram. If required, you can create a new folder within that directory to store your customized diagrams, rather than storing them in the \ud folder.

NOTE: This \ud directory should be located on a networked drive where it can provide multiple users with access to saved user diagrams.

> **NOTE**: You should notify users of the Diagrams application in the Web Applications component of the location of the new custom network diagram if they want to use it instead of the automatically generated diagram. Users of the Diagrams application need to update the registry settings on their computer to specify the location of the new custom network diagram. See "Custom network diagram setup" in the Diagrams section of the Web Applications online help.

Editing a user diagram

To edit a user diagram, the Diagram Objects toolbox must be displayed. To open the toolbox, select **Options > Show Toolbox**. If the **Show Toolbox** option is grayed out, your user account does not have sufficient access; contact your system administrator.

The Enable Undo option

The Enable Undo option enables or disables the undo feature in Vista. With this option active, you are allowed to undo or "step backward" up to three times. This feature is enabled by default. If you disable the undo feature, you lose the ability to undo any previous changes. You cannot enable the feature and undo changes made prior to enabling the feature. A check mark appears beside Enable Undo in the Options menu when this option is selected.

> **NOTE**: The **Show Toolbox** option must be selected for the **Enable Undo** option to become active — it remains unselectable otherwise.

Adding a diagram object to a user diagram

You can add a new diagram object to a user diagram by dragging it from the Diagram Objects toolbox.

Types of diagram objects

Diagram objects provide the interface to ION registers in your network. Each diagram object performs a single function. For example, a single numeric object placed in a user diagram displays numeric data from a single source. There are seven types of diagram objects available on the Diagram Objects toolbox:

Grouping Object:

Group a number of related objects within a separate grouping window. Click on a grouping object to open the grouping window and display its contents.

Text Box:

Display independent headings, titles, and comments in a user diagram.

Numeric Object:

Display real-time numeric values such as Volts, Amps or kW.

Status Object:

Display Boolean data (On/Off, Yes/No, 1/0, etc.) or device-dependent conditions to indicate the status of a digital output (relay), digital input (status input) or setpoint. The image can be customized and it can be configured to use animation.

Data Log Viewer:

Display data logs stored in the ION_Data database (including archives from previous versions of the product), show waveform data (waveform capture, waveform recorder) and trends (data recorders) using tables, and provide data for plotting.

Event Log Viewer:

Display event logs stored in the ION_Data database (including archives from previous versions of the product), show event records from devices and other components, and display alarms.

Control Object:

Send commands to a device on your network (for example, clear an energy accumulator, toggle a relay, reset a counter, or adjust the value of an analog output device).

To add a function to a user diagram, choose the type of diagram object you want, drag it into the user diagram then link it to the appropriate register in a node. The following sections explain how to add and link a diagram object.

See Monitoring your system in Vista for more information on the function and basic use of each type of diagram object.

Adding a diagram object to a user diagram

1. Select **Options > Show Toolbox** if the toolbox is not displayed. The toolbox appears in the workspace.

2. Click on the diagram object you want and drag it from the toolbox to the diagram.

   In most cases, the object appears in the window as the icon from the toolbox. However, there are two exceptions:

   – A **text box** appears in the window as a rectangular box that displays the default message: "Your text goes here."

   – A **numeric object** appears in the window as a rectangular box with a colon (:). This indicates that the object is not yet linked to a node.

   The object is now ready to be linked to a real-time or logged data source. See Linking a diagram object to a data source for details on linking an object to a source.

NOTE: When a diagram object is placed in a window, it automatically uses the same node as the window. If you want to change this source, either relink each diagram object or change the default window link before placing the objects.

Cutting, copying, and pasting a diagram object

Use the Cut, Copy and Paste commands on the Edit menu to make a copy of a diagram object and paste it into another window or user diagram.

You can paste the copy as many times as you like to create multiple versions of an object. Each copy retains all of the attributes and linkage information of the original.

Cutting or copying a diagram object

1. Select the object(s).

2. Do one of the following:

   - To remove the original, select **Edit > Cut** then click **Yes** to confirm the deletion or **No** to cancel.

   - To retain the original, select **Edit > Copy**. The selection is copied to the WinPM.Net clipboard.

NOTE: The WinPM.Net clipboard is a temporary storage area that holds any information cut or copied from Vista. It should not be confused with the Microsoft Windows Clipboard.

After the object is copied to the clipboard, you can paste it into the active window.

Pasting a diagram object

1. Select the window you want to paste to.

2. Select **Edit > Paste**. The object appears in the active window.

Copying a group of diagram objects to a framework

A Vista framework is a template that you can use to record the layout of a group of diagram objects.

When you select **Edit > Copy to framework**, Vista saves the selection (including the type and location of each diagram object) as a file. You can then use the Paste from framework command to copy the group layout to another diagram or grouping window. This is useful for replicating a single configuration on multiple workstations. You can store a saved framework with its own filename and retrieve it anytime.

Copying a group of diagram objects to a framework

1. Select the group of objects you want to copy.

2. Select **Edit > Copy to framework** to open the **Copy to framework** dialog.

3. Type a name for the framework in the File name box then click **Save**. The framework file extension (.fwu) is added automatically.

   The selection is saved as a framework.

After you save a group of objects as a framework, you can paste it to another window.

Pasting a group of diagram objects from a framework

1. Select the window you want to paste to.

2. Select **Edit > Paste from framework** to open the **Paste from framework** dialog.

3. Navigate to the framework you want to paste and click **Open**. The framework selection appears in the active window.

Adjusting diagram object layout and size

This section describes how to move, align and resize diagram objects.

Moving diagram objects

You can move a single diagram object or a group of diagram objects within a single window or from one window to another.

1. Select the object(s) you want to move.

2. Drag the selected object(s) to the new location.

You can also use the arrow keys on the keyboard to move the selected diagram object(s) one space at a time.

When diagram objects overlap, use the Send to Back command to place one object beneath the others. Select the object then select Edit > Send to Back (or press CTRL+K).

Resizing a single diagram object

You can adjust the size of a diagram object to make it larger or smaller or you can stretch it horizontally or vertically to change its shape.

1. Click on the diagram object to select it.

2. Do one of the following:

   – To make the object larger or smaller, drag a corner handle.

   – To stretch the object horizontally or vertically, drag a middle handle.

Resizing a diagram objects to a standard size

The **Size** tab in the Layout dialog lets you automatically resize a diagram object or group of diagram objects to a set of standard dimensions.

1. Select the diagram objects you want to resize.

2. Select **Edit > Layout** or click to open the **Layout** dialog.

3. Select the **Size** tab.

4. Edit the horizontal and vertical dimensions of the object(s) as required.

   – Select **No change** to make no changes to that dimension.

   – Select **Shrink to smallest** or **Grow to largest** to adjust the width and/or height to match that of the smallest or largest object in the selected group.

   – Select **Pixels** to enter the exact width or height in pixels.

   You can also combine horizontal and vertical options on a single selection. For example, if you select Shrink to Smallest in the Horizontal section and 30 pixels in the Vertical section, the entire selection is resized to the width of the shortest object and the specified height of 30 pixels.

5. Click **OK** to save your changes.

Aligning diagram objects relative to each other

The **Align** tab of the Layout dialog lets you specify the horizontal and vertical arrangement of the selected objects.

1. Select the diagram objects that you want to align.

2. Select **Edit > Layout** or click ![icon] to open the **Layout** dialog.

3. Select the **Align** tab to specify the horizontal and vertical alignment of the object(s).

    The options in the Horizontal and Vertical areas determine the criteria by which you can align diagram objects. For example, if you select **Left sides** under Horizontal and **Space evenly** under Vertical, Vista aligns all objects in the selection by their left sides (using the leftmost object for reference) and distributes them evenly along a vertical axis.

4. Select the options you want then click **OK**.

Aligning diagram objects on a grid

The **Grid** tab of the Layout dialog lets you activate and configure an invisible grid. Objects placed onto the grid snap to the nearest gridline to help align and space them evenly.

To activate or deactivate the grid:

1. Select **Edit > Layout** or click ![icon] to open the **Layout** dialog.

2. Select the **Grid** tab.

3. Select one of the following options:

    −   **No grid** turns the grid off.

    −   **Grid size in pixels** activates the grid. Type a number in the "Grid size in pixels" box to specify the size of the grid (distance between grid lines).

4. Click **OK** to save your changes.

Customizing a diagram object

After you have placed and linked a diagram object, you can customize it. Right-click a diagram object in edit mode to display the configuration options available for that object type.

Customizing a diagram object

1. Right-click the diagram object to open the **<Object> Configuration** dialog (where <Object> is the type of diagram object selected).

2. Select the appropriate tab and make your changes. Repeat this step for the other tabs until you have made all your desired changes.

3. Click **OK** to save your changes to the diagram object.

The options available vary depending on the tab and the object selected. See the sections below for details on the options available for different tabs and objects.

The Preview pane

Use the Preview pane to view how your proposed changes will appear in the diagram.

For some tabs, you access dialog boxes by right-clicking on the image in the Preview pane (for example, the Display tab of a numeric object). More information is given in the relevant sections that follow.

Available configuration options

The following table lists the various configuration options available for each type of diagram object.

Available Option Tabs	Diagram Objects						
	Grouping Object	Text Box Object	Numeric Object	Status Object	Data Log Viewer	Event Log Viewer	Control Object
Caption	x		x	x	x	x	x
Text		x	x	x			x
Edit Text		x					
Link (or Node)[1]	x		x	x			x
Query Server[2]	x		x	x	x	x	x
Display	x		x	x			x
Action	x		x	x			x
Box		x	x				
Annunciation				x			
Query[3]					x	x	
Alarming						x	
[1] Node is the tab available with Grouping Objects.							
[2] Query Server is only available on Grouping, Numeric,Status, and Control objects when the Action is set to something other than none.							
[3] See Querying the database for information on configuring this option.							

Caption options

To view caption options, right-click the object then select the **Caption** tab.

Use the **Caption** tab to add custom, user-defined text to a diagram object. The Caption tab is available for the following diagram objects: Grouping, Numeric, Status, Data Log Viewer, Event Log Viewer, and Control.

The Caption tab contains three sections: **Caption**, **Font**, and **Position**.

The **Caption** section determines whether a diagram object displays a textual comment along with the diagram object's icon. This section has three options:

- Select **None** to display no caption.
- Select **Use default** to display the default Vista caption (i.e., the device name or data source).
- Select **Use custom** to display the text that you type into the field provided.

The **Font** section allows you to change the font used in a caption. This section has two options:

- Select **Inherit from parent window** to use the default font.

- Select **Custom** to specify a custom font. Click **Font** to open the **Font** dialog and change the font, size, style or color.

The **Position** section has two options:

- Select **Top** to place the caption above the diagram object.

- Select **Bottom** to place the caption below the diagram object.

Text options

Some diagram objects have a text label integrated into the display. To edit the appearance of this label, use the options offered on the Text tab. The Text tab is available for the following diagram objects: Numeric, Status, Control, and Text.

> **NOTE**: You can edit the text displayed in a text label. For a numeric, status or control object, use the **Label** section of the **Link** tab. For a text box object, use the **Edit Text** tab.

To view the Text options, right-click the object then select the **Text** tab. The **Text** tab has two sections: Font and Position.

The **Font** section has two options:

- Select **Inherit from parent window** to use the default font

- Select **Custom** to specify a custom font. Click **Font** to display the **Font** dialog and change the font, size, style or color.

The **Position** section has three options — **Left**, **Center** and **Right** — to define the alignment of the text in relation to the diagram object. Use the Preview area located beside the Text tab to view your proposed changes.

Edit Text options

To view the Edit Text options, right-click the object then select the **Edit Text** tab.

Use the **Edit Text** tab to specify a text message for the text box. The Edit Text tab is available on the text box object only.

The **Edit Text** tab has two options:

- **Use default** displays the window link information (i.e., the node).

- **Use custom** displays your custom text message.

If you select custom, use the text box to enter your text. The text box supports multiple lines of text. If your message appears incomplete or truncated after you enter text and click **OK**, you can adjust the dimensions of the text box object. To resize a text box, select it then drag its handles.

The font used in the text box is based on the default setting specified for the active window. Use the Text tab to specify a different font or position for the text box. Use the Box tab to specify a border style or change the background color.

Link options

To view the link options, right-click the object then select the **Link** tab.

Use the **Link** tab to link a diagram object to a real-time source, and to specify a new text label. The Link tab is available for the following diagram objects: Numeric, Status, and Control.

> **NOTE**: To set a real-time source for a grouping object, use the Node tab.

The Link tab contains two sections: Link and Label.

Use the **Link** section to specify a real-time source for the diagram object. See Linking a numeric, status or control object for more information.

Use the **Label** section to change the object's text label. The Label section has three options:

- Select **None** to display no label.

- Select **Use node label** to display the node's register label. This is the default.

- Select **Custom** to display the label that you type into the text field provided. The custom labels you create are stored in the user diagram and are only available to users of your diagram.

> **TIP**: If you have specified custom labels in your diagram, or custom labels have been applied to an ACCESS meter at the device level, you can display the default labels at any time by selecting **Show Default Labels** from the **Options** menu. This displays the default label of the data source instead of the label assigned in the device or in the user diagram.

Query Server options

Use the Query Server tab to connect a diagram object to a Query Server. The Query Server tab is available on the Data Log Viewer and Event Log Viewer objects. See Linking a Data Log Viewer or Event Log Viewer for more information.

The Query Server tab is also available on grouping objects and any object configured to open a grouping window or user diagram. In this case, use the Query Server tab to specify a default Query Server for the associated grouping window or user diagram. See Query Server: setting a default Query Server for a window for more information.

Display options

Use the **Display** tab to customize the appearance of a diagram object. The Display tab is available for the following diagram objects: Numeric, Status, Control, and Grouping.

The Display tab offers a unique set of options for each type of diagram object. To view the display options, right-click the diagram object then select the Display tab.

Customizing the display of a numeric object

Numeric objects display real-time data in your user diagram. The default numeric object appears as a rectangular alphanumeric display with a single decimal place of resolution and no unit symbol. You can customize the display to change any of these default elements.

NOTE: Vistasupports a maximum of 59 non-alphanumeric numeric objects within a single window. To change the way numbers are displayed (thousands separator and decimal point), use the Regional Settings option of the Windows Control Panel. You can also use this option to customize the time and date styles. (The year/month/day format can be modified, and the clock can be changed from am/pm to 24 hour.)

Setting Display style

The **Display style** box lets you select a new style from the list of available choices. The following styles are available:

- Alphanumeric: Default style. General-use text display.

- Partial Dial: General use voltage, current, harmonic distortion, etc.

- Full Dial: power factor, bi-directional power flow, etc.

- Horizontal Bar: General use.

- Vertical Bar: General use.

- Scrolling Graph display: General use. Displays recent readings on a scrolling graph.

- Arc Meter Display: General use voltage, current, harmonic distortion, etc.

- Timestamped display: Displays the value, and the date and time the value was measured by the device. Useful for min./max. values.

- Timestamp Only: Same as timestamped, but does not display a value. Can be used as a clock.

- PC Timestamped Display: Displays the value, and the date and time the value was received by Vista.

Setting Units and Resolution

Use the **Units** box to type the unit name for the measured value (for example, "Volts" or "Amps"). This name is displayed on the numeric object.

In certain cases, the value is scaled. For example, if the unit name begins with K, M, or G (for example, kV or MVolts), the displayed value is adjusted accordingly. However, if the unit contains certain letters (i.e., kW or kVA), the scaling factor is reduced by 1,000 (for example, if the unit begins with K, the value is not scaled and M is scaled by 1,000). The reason for this is that most devices provide these values already scaled.

Use the **Resolution** box to specify the number of decimal places to be displayed for a numeric object. The default value is 1, but this can be increased to take advantage of any extra resolution available from the data source.

NOTE: Specifying higher resolution does not result in higher accuracy than the specified accuracy of the device.

Setting the Font options

Use the **Ticks Labels Font** section to define font options (such as style, size, and color) for the numeric object. This section has two options:

- Select **Inherit from parent window** to use the default style.

- Select **Custom** to specify a custom style, then click **Font** to open the **Font** dialog and make your changes.

Setting Flags

Use the **Flags** button to display the Normal Operating Range dialog. Use this dialog to set a Low Low Limit, Low Limit, High Limit, and High High Limit, and define a normal operating range to be displayed on the numeric object. After flags have been set, you can use them to quickly determine whether a reading is within the normal range you defined.

The effect of setting these limits depends on the display style you have specified. For example, on the Partial Dial, Full Dial, and Arc Meter displays, flags appear as colored ranges on the dial. On other types of numeric objects, the flag function changes the object's background color whenever a limit is exceeded.

To view or change the default colors, select **Options > Flag Colors**.

NOTE: The flags used in Vista are not linked to setpoints in ACCESS meters. Setpoint activity is automatically logged by the Log Inserter. Setpoint status can be monitored using a status object and/or data log viewer.

Other display options for Vertical or Horizontal Bar graphs

In the Preview section of a vertical or horizontal bar graph Display tab:

- Right-click the axis labels to display the Axis Labels dialog. Select the Format (Decimal, Scientific, or Engineering) and determine the Axis Title (none or custom text).

- Right-click the axis tick marks to display the Vertical or Horizontal Axis dialog. Configure the Range, Ticks and Grid style. Select Logarithmic scale to enable the logarithmic scale option.

Other display options for Partial Dial, Full Dial and Arc Meter displays

To configure meter parameters on partial dial, full dial or arc meter displays, right-click anywhere in the Preview section of the Display tab.

Set the Range, Arc Position, Ticks, and dial Position. For full and partial dial displays, set the label position to inside or outside. Click Line Attributes to set the style and width of the dial.

Other display options for Scrolling graphs

In the Preview section of the Display tab:

- Right-click the vertical or horizontal axis tick marks to open the Vertical and Time Axis dialogs. Set the Range and Ticks, and (for the vertical axis) Grids Style.

- Right-click the vertical axis labels to display the Axis Labels dialog. Select the Format (Decimal, Scientific, or Engineering) and determine the Axis Title (none or custom text).

- Right-click the graph background to display the Scrolling Graph dialog where you can set the Sample Interval and Relative Reset.

The sample interval determines the resolution of the X-axis in seconds. It should be set between one tenth and one hundredth of the X-axis range; for example, when extending the range to one hour (3600 seconds), the sample interval should be set to approximately 60 seconds.

The relative reset is a percentage that determines the amount the scrolling graph shifts back upon filling the plotting area. (For example, a relative reset of 0.5 shifts back 50%; a relative reset of 0.99 scrolls the most smoothly.)

Customizing the display of a status object

Status objects display the status of Boolean "on/off" conditions in one of three ways: an indicator light, an alphanumeric field, or a custom image.

To view the display options of a status object, right-click the object then select the **Display** tab.

> **NOTE**: To invert the default color scheme, select "Invert active state" from the Annunciation tab.

The Display tab has three options:

- Select **Default image** to display the default indicator light image, which changes color to indicate various conditions: by default, green indicates an OFF condition, red indicates an ON condition and gray indicates a status object that is not connected or is not receiving information. Device dependent multi conditions (HighHigh, High, Normal, Low, and LowLow) also use colors to indicate the various conditions.

- Select **Custom images** to use custom images for each condition. Custom images can show animation, customized buttons, and so on.

  For each item selected under **Condition** (ON, OFF, and Unconnected), click **Browse** to locate and select the image you want to associate with the condition. To hide the status object in a given state, leave the image field blank.

  Currently, the custom image option supports the following graphic file formats: BMP, animated GIF, transparent GIF and PNG, WMF (Windows Metafile), EMF, JPG, AVI (without audio and uncompressed or compressed using RLE8 compression).

  > **NOTE**: GIF, JPG, and PNG images need to be in `\config\diagrams` under the product's installation location or they will not be shown in the Web-based Diagrams application.

  You can use three different files to represent the three display states (ON, OFF, and Unconnected) of a status object. Each of these files can also be of a different type (for example, BMP, GIF, and JPG), however you cannot mix AVI and animated GIF images within a status object.

- Select **Alphanumeric** to display the Boolean state or multi-condition state labels stored in the device and choose custom colors to visually indicate the corresponding states.

  For example, suppose "Running" has been stored in a meter as the label for the Boolean register that stores the ON condition of a motor relay and "Stopped" for the OFF condition. You can show this register label together with the status label using the alphanumeric display style. You can override the meter labels on the Link tab.

  Click **ON Color** and **OFF Color** to specify the colors associated with each condition.

You can also have multi conditions if you link the status object to an enumerated register instead of a boolean. You can do this for BCPM devices.

Customizing the display of a grouping object

The grouping object can be displayed as the default image, a custom image, or no image. You can customize the display to change any of these default elements.

To view the display options of a grouping object, right-click the object then select the **Display** tab.

The Display tab has three options:

- Select **Default image** to use the default image for a grouping object that appears in the toolbox.

- Select **Custom image** to display a custom image. Enter the filename of the image you want to use or click **Browse** to locate and select an image. The filename should include the file's extension (.bmp, .jpg, .png, .gif, .emf, .wmf, or .avi).

- Select **None** to hide the grouping object. With no image specified, the grouping object is transparent unless you select it (displaying the handles). If you move the mouse over a transparent object, the status line displays the size and object type (for example, "30x30:Grouping Object" in the status line means the hidden object is a grouping object and its size is 30 pixels by 30 pixels).

  Hiding a grouping object is especially effective when using a custom background image; you can integrate your transparent grouping objects into the image by placing them at the appropriate points on the drawing. Users can display site-specific information by double-clicking on the appropriate spot in the background. See Image: Displaying a background image for information on using a custom background.

Customizing the display of a control object

A control object appears differently depending on the type of register to which it is linked. This same criteria also determines the display options available.

To view the display options of a control object, right-click the object then select the **Display** tab.

The options available depend on the type of control object.

External Numeric

A control object that is linked to an External Numeric module appears as an alphanumeric display box. The Display tab has the following sections: Border and Background Color.

In the **Border** section, select Hide to show no border or select Show to display a border. If you select Show, you can specify the width in pixels and click Color to choose a color for the border.

In the **Background Color** section, select Transparent for no background or select Custom then click **Color** to define a color for the background.

External Pulse

A control object that is linked to an External Pulse module appears as a button icon, by default. The Display tab has three options: Default image, Custom image and None.

Select Default image to display the button icon. Select Custom image and click **Browse** to select a different image. Select None to show no image (the control object is hidden but still functions).

External Boolean

A control object that is linked to an External Boolean module appears as a switch icon, by default. The Display tab has two options: Default image or Custom images.

Select Default image to display the switch icon. Select Custom images then click **Browse** to define custom images for ON, OFF and Unconnected. To hide the control object for a given state, leave the appropriate box blank. Currently, the custom image option supports the following graphic file formats: BMP, JPG, PNG, GIF, EMF, WMF (Windows Metafile), and AVI (without audio and uncompressed or compressed using RLE8 compression).

Note that you cannot mix AVI and animated GIF images within a control object.

Action options

To view the action options, right-click the diagram object then select the **Action** tab.

Use the Action tab to specify the response of a diagram object when it is clicked (in Display mode) or double-clicked (in Edit mode).

The Action tab is available for the following diagram objects: Numeric, Status, Grouping, and Control objects. Depending on the type of diagram object, the Action tab displays one of two sets of options:

- Specify an action for a numeric, status, or grouping object

- Customize the action for a control object

Specifying an action for numeric, status, or grouping objects

NOTE: If you change the action for a grouping object that is already associated with a grouping window, the existing window (and any objects within it) is deleted. Vista displays a message before performing this action.

When you right-click a numeric, status, or grouping object, the **Action** tab displays the available options:

- Select **None** if you do not want the object to support an action. This is the default for numeric, status, and grouping objects.

- Select **Open Grouping Window** to open a grouping window when the object is clicked. You can use this grouping window to display additional information. For example, if a numeric object displays a total value (such as total kW), you can display specific information (such as kW readings for each phase) in an associated grouping window.

- The **Open Diagram for Meter Template** option is used by default each time a network diagram is generated. Each meter icon in the network diagram is set to open the default diagrams that match that meter's configuration.

- Select **Open User Diagram** to open a user diagram when the object is clicked. Type the full path and filename into the field provided or click **Browse** to locate the diagram you want. Using this action, you can choose a diagram object that displays a representative reading from an independent diagram and use it to open the other diagram when clicked.

- Select **Open File** to open a file when the object is clicked. Type the full path and filename into the field provided or click **Browse** to locate the file you want. The file can be a third-party program or some other document; if a document is specified, then the program associated with the file extension is launched and the specified document is loaded into the application.

- Select **Open Web Page** to open a specified webpage in your default web browser when the object is clicked. Type the entire address into the field provided.

NOTE: When you select any action other than None, the Query Server tab is added to the top of the object configuration box. You can use this tab to specify a source of logged data for the new user diagram or grouping window. If you enter only the filename into the field provided (and do not specify a directory), Vista defaults to
...\WinPM.Net\config\diagrams\ud.

Customizing the action for a control object

To view the action options of a control object, right-click the control object and select the **Action** tab.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software to control time-critical functions.
- Do not use the software to control remote equipment without proper access control and status feedback.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

The Action tab offers three options that you can use to define how the control object responds when clicked:

- Select **No confirmation** to have no confirmation message displayed.

- Select **Message box confirmation** to display a message box when the object is clicked. The message box requires the user to click either **OK** or **Cancel**.

- Select **Password confirmation** to request the user's logon password when an object is clicked.

The **Level to operate** list allows you to specify the access level required to perform the action.

Box options

To view the Box options, right-click the object then select the **Box** tab.

Use the Box tab to add a border or background color to a diagram object. The Box tab is available for the following diagram objects: Numeric, Text Box, and any status object that has been configured as an alphanumeric display

The Box tab has two sections:

- **Border** lets you adjust a border's width and color as well as whether or not it is displayed. Select **Hide** to remove the border or **Show** to display it. If you select Show, click **Color** to define a border color and type a number in the **Width in pixels** field to change the line thickness.

- **Background Color** lets you set the background color of an object. Select **Transparent** to use the background color of the parent window or select **Custom** and click **Color** to select another color.

> **NOTE**: A Numeric Object uses the diagram background color in Vista when set to **Transparent**, which makes it look transparent. In Diagrams (web application), this same Numeric Object is displayed as truly transparent. Text Box Objects are truly transparent in Vista and Diagrams when set to **Transparent**.

Annunciation options

To view the annunciation options, right-click the Status Object to open the **Status Object Configuration** dialog, then select the **Annunciation** tab.

Use the Annunciation tab to specify how users are notified of a change of status. The Annunciation tab is available only on the Status Object.

> **NOTE**: The Annunciation feature requires that the Status Object's diagram is displayed; if the display is closed or minimized, then the annunciation feature does not work.

The Annunciation tab has three options that set the action that Vista performs when a status object becomes active:

- Select **Command line** to automatically launch another application in the event of a change in status. Type the program's executable (.exe) command into the field provided or click **Browse** to locate the appropriate file.

- Select **Message box** to specify an annunciation message. Type your message into the field provided. The first line of text appears as the title of the message box.

- Select **Invert active state** to change the way the status object reacts to an active state. When this option is selected, the status object treats an active state as inactive and vice-versa. This is useful, for example, in cases when you want to be notified if a function (such as a cooling fan) switches off.

> **NOTE**: If you select the Invert active state check box, this also inverts the default colors of the status object.

Alarming options

Every type of event that occurs in a WinPM.Net network has a prioritized value that identifies its relative significance on a scale of zero to 255. By default, Vista identifies any event with a priority of 128 or greater as an alarm.

To view the Alarming options, right-click the Event Log Viewer then select the **Alarming** tab.

Use the **Alarming** tab to create or delete a priority range, specify the security level required to acknowledge an alarm, or customize the way Vista annunciates an alarm.

The Alarming tab has three main sections:

- **Priority ranges** lists the currently configured alarm ranges. When you select a range in the Priority ranges section (for example "Range 192-255"), the other sections (Level to Acknowledge and Annunciation) display the current settings for that range. You can proceed to modify any of these settings for the specified range. (Your changes affect only the currently selected priority range.)

  Click **New** to add a new priority range. In the **Priority** dialog box, type a number representing the lower limit of the new range then click **OK**. The upper limit of the new range is automatically defined by the lower limit of the existing range. For example, if you have an existing range of 128-191, you can type 100 then click **OK** and the range 100-127 appears. To delete an existing range, select it and click **Delete**.

- The **Level to acknowledge** list specifies the access level required to acknowledge alarms in the selected range.

- **Annunciation** lets you customize the way Vista annunciates an alarm:

    - Select Beep to play a beeping sound.

    - Select Flash to have a flashing display.

    - Select Command line to have Vista to automatically launch another application in the event of an alarm. Type the program's executable (.exe) command into the field provided or click **Browse** to locate the appropriate file. This option can be used to automatically alert other users of specific events through paging systems, fax transmissions, or any other command line capable application.

    - Select Message box to display an annunciation message. When you select this option, type your message into the field provided. The first line of text appears as the title of the message box and the remaining text appears as the content of the message box.

By default, Vista uses a beeping sound to annunciate an event with a priority between 128-191 and a beeping sound combined with a flashing display to annunciate an event with a priority between 192-255.

NOTE: If your computer is equipped with a sound card, you can replace the default alarm sound with a custom sound. Save the sound file you want to use as "alarm.wav" and store it in the ...\WinPM.Net\system\etc directory.

Linking a diagram object to a data source

After placing an object in your user diagram, you need to link the object to a data source. Some types of diagram objects require a real-time data source and others require a logged data source:

- Numeric, status, and control objects require a real-time data source. You can link these diagram objects to a device.

- Event Log Viewers and Data Log Viewers require a logged data source. You can link these diagram objects to any log in your system.

Linking a numeric, status or control object

This section describes how to link a numeric, status, or control object to a node. Numeric, status, and control objects display real-time data. After you have placed one of these objects into a user diagram, you should link it to a real-time data source. An object cannot receive or display information until it is linked.

Specifying a real-time link to a node

1. Right-click the diagram object you want to link to open the **Configuration** dialog.

2. Select the **Link** tab.

   The Link tab contains two sections: **Link** and **Label**. Use the **Label** section to define a label for the diagram object (see Link options for information on the Label section) and use the **Link** section to choose a real-time source.

3. Select one of the following:

   - **Inherit from parent window**: The selected diagram object defaults to the same node used by the parent window (the window in which the object resides). You must specify a real-time register within the node, but the node itself is pre-selected.

   - **Custom**: Allows you to select a new node. Select **ION** to choose a different device than that used by the parent window.

4. Click **Edit Link** to open the **Create Link** dialog.

5. Specify a node, manager, module and output register:

   a. Double-click the node in the Nodes box that contains the data source you want. If you chose Inherit from parent window in step 3, the node is preselected.

   b. Double-click the manager you want (for example, Power Meter modules) from the Managers box.

   c. Double-click the module you want from the Modules box (for example, Power Meter).

   d. Double-click the output register you want from the Output Registers box (for example,Vll avg).

6. Click **OK** in the **Create Link** dialog to save your changes.

   The **Create Link** dialog closes and the new link information is displayed along the lower edge of the **Configuration** dialog.

7. Click **OK** on the **Configuration** dialog to save your new configuration.

Linking a Data Log Viewer or Event Log Viewer

The Event Log Viewer and Data Log Viewer display logged data. After you have placed an Event Log Viewer or Data Log Viewer object into a user diagram, you need to link it to the WinPM.Net Query Service.

Linking a diagram object to the WinPM.Net Query Service

1. Right-click the Event or Data Log Viewer object you want to link.

2. Select the **Query Server** tab in the **Configuration** dialog.

   The **Query Server** tab has two options:
   - Select **Inherit from parent window** to have the selected diagram object default to the same Query Server used by the parent window (the window in which the object resides). If you select this option, click **OK** to exit, skip the following steps, and continue with Querying the database.
   - Select **Custom** Query Server and click **Edit Link** to open a dialog where you can select a new Query Server link.

3. Select **Custom** Query Server and click **Edit Link** to open the **Create Link** dialog.
   - Use this dialog to specify the Query Server, Query Manager, Query module, and Query register.
   - Double-click the Query Server that you want from the Nodes box.
   - Double-click Query Modules from the Managers box.
   - Double-click the Query module that you want from the Modules box.
   - Double-click the Query Register that you want from the Output Registers box.

4. Click **OK** the **Create Link** dialog to save your changes.

   The name of the selected WinPM.Net Query Service is displayed as the Group Name on the Query Server tab of the **Configuration** dialog.

5. Click **OK** on the **Configuration** dialog to accept your new configuration.

After you have linked a diagram object to a log server, you need to define a query to specify the type(s) of logged data that you want the object to display. See Querying the database for more information.

> **NOTE**: If the Query Server has multiple Query modules that are accessing multiple databases, you need to select the Query module that is connected to the database that contains the data of interest.

Querying the database

The Data Log Viewer and the Event Log Viewer query the database, extract logged data, and display that data in Vista.

A query is a set of instructions that the log viewer uses to request particular data from the database. Each log viewer has its own individual query. Editing this query information changes what data the query retrieves and the way a log viewer displays that data.

Right-click the data or event log viewer object and select the Query tab to view the query options or access the Query Wizard. Use the query options to define the way a viewer displays data and the Query Wizard to edit the actual query.

> **NOTE**: Anytime you place and link a new Data Log Viewer or Event Log Viewer, you need to specify a query for it. You can specify a query using the Query Wizard.

Using the Query options

To define the way a log viewer displays information, configure its query options.

To view the query options for an Event Log Viewer or Data Log Viewer, right-click the viewer object to open the **Configuration** dialog. Select the **Query** tab.

The upper section of the **Query** tab offers two buttons:

- **Edit Query** starts the Query Wizard, a user-friendly interface that helps you edit the query.

- **Edit SQL** accesses the SQL statement via Windows Notepad so you can edit the query directly using SQL, if preferred.

> **NOTE**: If you edit a query directly (i.e., by editing the SQL statement), your changes may not be accessible to the Query Wizard. This is not a problem unless you intend to use the Query Wizard as well.

The **Options** section of the Query tab offers the following options:

- **Records uploaded at a time**: This option specifies the number of records Vista uploads into active memory (RAM) when you first open a Data or Event Log Viewer. The maximum value is 9999. As you scroll down the list of records, Vista uploads additional records as needed. You may want to change this default value if, for example, you want to select a large number of records for plotting, without scrolling down repeatedly.

- **Update automatically**: This option is disabled (cleared) by default. The Data or Event Log Viewer does not display any new records; rather, the Log Viewer initially uploads the specified number of records (for example 100) and does not update again unless it is closed (and its query results deleted) then reopened.

  With this option selected, the Log Viewer continually receives and displays new records up to a maximum of 1000 records; the Log Viewer checks for new records every 20 seconds. However, the updates can consume a noticeable amount of CPU power, especially if several Log Viewers remain open simultaneously or if Log Viewers are uploading waveform data.

- **Delete query on close**: This option is enabled (selected) by default. The Log Viewer clears its temporary records from active memory (RAM) upon closing so that every time a Log Viewer is opened, the database is queried again and all available records are displayed. (This option does not affect any original records stored in the database.)

  If this option is disabled, Vista caches the record set returned to the Log Viewer, meaning that if the Log Viewer is closed then reopened, the table is populated from memory and not from the database. Any new records inserted into the database during the time the Log Viewer was initially open do not appear.

> **NOTE**: Since it is recommended that **Update automatically** remain disabled in all but the most compelling circumstances, it is important that the **Delete query on close** option remains enabled.

- **Use parent node**: With this option selected, the log viewer uses the same node linkage as the parent window. The node is also pre-selected in the Query Wizard - you should not select this option if you intend to query more than one node.

X-parameter for Log View Plotter

For a Data Log Viewer, the X-parameter for Log View Plotter section is offered at the bottom of the Query tab. Use this section to specify a column to be used on the X-axis of the Log View Plotter. This section offers two options:

- **Timestamp** is selected as the X-parameter by default. In some cases, you may need to specify a different X-parameter. For example, when you are plotting disturbances, you need to plot your data against the Duration column.

- **Use column #** lets you specify a new x-parameter. In the field provided, type the column number that you want to use as the X-parameter (for example, 3). When you click **OK** on the Data Log Viewer Configuration box, the specified column changes to blue.

> **NOTE**: In the Data Log Viewer, the column specified for the X-parameter is colored blue for easy identification.

Using the Query Wizard

The Query Wizard helps you edit an existing query or create a new query for a Data Log Viewer or Event Log Viewer.

1. Right-click the log viewer that you want to configure.

2. Select the **Query** tab in the Configuration box.

3. Click **Edit Query** to open the Query Wizard.

   The Query Wizard consists of four steps designed to guide you through the process of editing a query. These steps are described in more detail in the following sections.

   - Selecting the node(s) to query.

   - Specifying logs within the specified node(s).

   - Specifying columns and specifying filters.

   - Choosing a sort order.

   Depending on the type of query (new or existing), you may not need to perform all four steps. When you start the Query Wizard, it automatically opens on the appropriate step in the process.

4. Use the four buttons at the bottom of the Query Wizard to move through or exit the Query Wizard:

   - Use the **Back** and **Next** buttons to navigate through the wizard.

   - Click **Cancel** to quit the Query Wizard and discard your changes.

- Click **Finish** to quit the Query Wizard and save your changes.

5. When finished, click **OK** to save your changes and close the viewer configuration box.

The following example describes each step in the process of creating a typical query: configuring a Data Log Viewer to display waveforms for three phases (V1, V2, V3).

Step 1: Selecting the nodes

The Available list displays the available nodes (device or software) and the Selected list displays the currently selected nodes.

Select a node (or nodes) for your query. If a node has already been specified or you select Use Parent Node on the Query tab, the Query Wizard proceeds directly to Step 2: Specifying logs.

1. Click **Edit Query** on the **Query** tab of the Log Viewer Configuration dialog to start the Query Wizard.

2. Highlight the name of the node that you want from the Available list and click **Add**. The node appears in the Selected list. Repeat this procedure to add additional nodes for this query, if required.

   If you want to remove a node from the Selected list, highlight the node and click **Remove**. The node disappears from the Selected list.

3. Click **Next** to continue.

NOTE: If you want to view or edit a node's SQL statement directly, highlight the node in the Selected list and click the **SQL** button. The Windows Notepad displays the SQL statement for the specified node.

Step 2: Specifying logs

Specify one or more data logs to query. If a valid query has already been specified, the Query Wizard proceeds directly to Step 3: Specifying columns.

1. The Available list displays the logs available on the specified node. The Selected list displays the currently selected logs.

2. Highlight the name of the log(s) that you want from the Available list and click **Add**. The log (s) appears in the Selected list.

   NOTE: Check for additional tabs. If you selected more than one node in step 1, this box requires you to specify logs for each node. Each tab is labeled "Node 1," "Node 2," etc. and is identified at the top of each tab. Select "Advanced log view" to access any logs whose configuration has changed over time. This check box enables access to old configurations of snapshot logs.

3. Click **Next** to continue.

Step 3: Specifying columns and filters

Use the **Select the columns to appear in the Log Viewer** section to add or remove a column from your log viewer and the **Filter** section to establish a range for your query and to screen out unwanted data.

If you selected multiple logs in step 2, this box contains a tab for each log. Specify the column and filter information for each tab.

Specifying columns to appear in the Log Viewer

The Available list displays the available columns and the Selected list displays the currently selected columns. The Selected list displays the columns in the same order (left to right) that they will appear in the log viewer (top to bottom).

To specify the columns to appear in the Log Viewer:

1. Highlight the name of the column(s) you want then click **Add**. The column(s) appears in the Selected list.

   Select the asterisk (*) at the top of the Available list to select all available columns.
   For example, if you want to select some basic columns for the first waveform log you chose in the last step, you could highlight timestamp, node, and V1 in the Available list.

   > **NOTE**: The * column option shows all inputs connected to a recorder for Data Log Viewers. For example, use the * option if you are using a framework and the configuration of the data log is different for several meters. The * option is not available if more than one table is selected, as in the example.

2. Repeat this procedure for each column that you want (and for each tab).

3. Adjust the order of the items in the Selected column—highlight the column name you want to move in the **Selected (in order)** list then click the up or down arrows to move it. The columns are listed from top to bottom in the order they will appear from left to right.

   For example, if you want the Timestamp column to be the first (leftmost) column in your log viewer, highlight "timestamp" in the Selected (in order) section then click the up arrow to move it to the top of the list.

After you have selected the columns you want, proceed to specify filter settings.

Specifying a filter

Use the Filter section to construct one or more filter statements for your log viewer. Filters are useful for restricting the scope of your query or specifying a priority range for alarms.

To edit an existing filter statement, double-click on it.

For example, if you want to instruct the log viewer to display data for the past month (excluding today), you could use the following procedure:

1. In the first dropdown list on the left, select "timestamp".

2. In the second dropdown list to the right, select "BETWEEN".

3. In the third dropdown list to the right, select "last_month_to_date()".

4. In the fourth dropdown list (on the right), select "today_12AM()".

5. Click **Insert**.
   The following filter definition appears in the area below the filter controls:
   "timestamp" BETWEEN last_month_to_date() and today_12AM()

If you wanted to instruct an Event Log Viewer to display only unacknowledged alarms, create a filter that only displays alarms that have no acknowledgement time (ack-time).

1. In the first box, select "ack-time"

2. In the second box, select "IS NULL"

3. Click **Insert**.
    The following filter appears in the box below the filter controls:
    "ack_time" IS NULL

As you construct a filter statement, each field offers options based on your selection in the previous field(s). For example, if you select "priority" in the first field, the third field becomes a text field where you can type a numeric value; if you select "timestamp" in the first field, the third field displays time-based options (i.e., now, today 12AM, yesterday, etc.).

The following table lists the time-related functions:

Function	Description
now_()	The current time.
today_12AM()	Today at 00:00:00.
yesterday()	Yesterday at 00:00:00.
last_week()	Seven days ago at 00:00:00.
this_month()	The first day of this month at 00:00:00.
last_month_to_date()	The same date of the month one calendar month ago at 00:00:00.
last_month()	The first day of the last calendar month at 00:00:00.
last_quarter_to_date()	The first day of the month three calendar months ago at 00:00:00.

Note that time-related functions only return one record. To return all of the records for the time period, you also need to specify `< today_12AM()`. For example, specifying a filter of `=yesterday()` returns only one record, but a filter specifying `>=yesterday()` AND `<today_12AM()` returns all of yesterday's records.

The second field offers a standard set of operators.

```
=
>
<
>=
<=
!=
LIKE
BETWEEN
IS NULL
IS NOT NULL
IN
```

Operator-specific data

If you select an operator that requires additional information, Vista displays a dialog where you can specify the required data.

For example, if you select the "node" column in the first field then select the "IN" operator from the second field, a dialog opens requesting that you specify the node you want to use.

If you specify any column other than "node" then select "IN", a box appears requesting numeric values.

After you have specified a filter, click **Next** to continue.

Step 4: Choosing a sort order

Specify a sort order. The sort order is the order in which records appear in the viewer.

1. Highlight the name of the column to sort by in the Available Columns list then click **Add**. The column appears in the Sort Order list.

   Available Columns displays the columns you chose in step 3.

2. Double-click on the column in the Sort Order list to select either an ascending or descending sort order.

   The Sort Order list displays the column(s) that dictate the order in which the records are sorted.

3. Click **Finish** to save your changes and quit the Query Wizard.

4. Click **OK** in the Configuration box to save your changes.

Customizing a diagram window

You can customize the appearance and configuration of any window within a user diagram. You can customize a window's font or background color, or configure window properties such as background image, real-time and logged data links, double-click action, and stale data settings.

In many cases, Vista uses the properties specified for the parent window as the default settings for any diagram objects or grouping windows contained within. By modifying a setting in the parent window, you automatically modify the same property on all objects inside the window that share this setting. (For example, any object with an option set to Inherit from parent window shares this information.) This can be useful when adding multiple diagram objects that are all linked to the same source.

To access any of these options, right-click anywhere in the background of the user diagram or window that you want to customize.

NOTE: Right-clicking in the background of a window displays a menu of available options; however, right-clicking on any object in the window while in Edit mode (including transparent objects such as text boxes or grouping windows) displays the configuration options for that object.

Default font

The font you select becomes the default setting for all captions and text labels within the window.

Changing the default font

1. Right-click the background of a user diagram and select **Default Font** to open the **Font** dialog.

2. Set the default font, style, size, effects, color, and format that you want.

3. Click **OK** to save your changes.

Background color

The default background color is gray. However, you can display any solid color as a background for any window in your user diagram.

Changing the background color

1. Right-click the background of a user diagram and select **Background Color** to open the **Color** dialog.

2. Select the color that you want or click **Define Custom Colors** for more options.

3. Click **OK** to save your changes.

Configuring window properties

To view the Window Properties Configuration options, right-click the background of any window in your user diagram and select **Properties**.

The Window Properties Configuration dialog opens with five option tabs: **Image**, **Query Server, Node**, **Action** and **Updates**.

Image: Displaying a background image

To view the image options, select the **Image** tab on the **Window Properties Configuration** dialog. Use the **Image** tab to specify a background image for the active diagram or grouping window.

The **Image** tab has two options:

*   Select **None** if you do not want to display a background image, or if you prefer to display a single color.

*   Select **Custom image** to display a background image. Enter the path and filename of the image you want to display. Vista supports images saved in the following formats: .bmp, .jpeg, .png, .gif, .emf, and .wmf.

You can create your own background image using a scanned photograph or a drawing created within a standard graphics program. Some ideas include:

*   An engineering drawing of your installation depicting the power distribution system and the location of critical elements.

*   A single line diagram illustrating the power distribution network or the individual components of your facility.

*   An aerial photograph of your facility depicting the approximate location of your monitoring equipment.

*   A satellite image or geographical map depicting your transmission and distribution network.

The following illustration depicts a sample Vista background with hidden diagram objects (here the object is visible by the handles because it is selected in Edit mode):
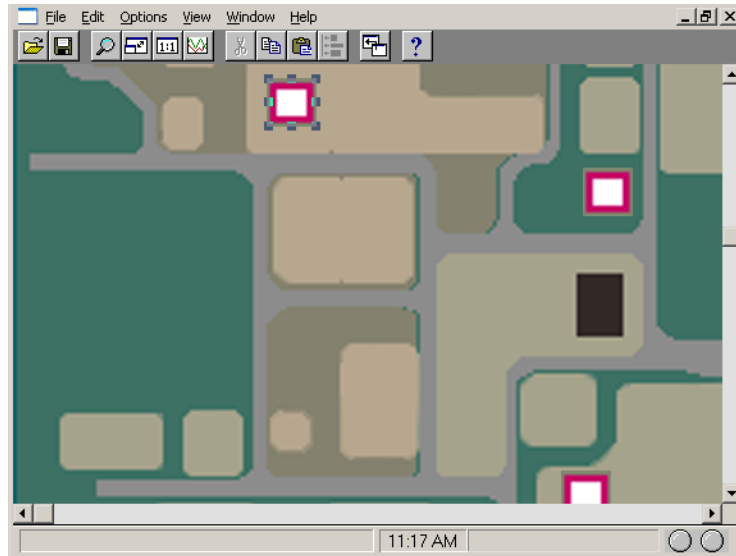
You can integrate your diagram objects into the image by placing them at the appropriate points on the drawing. Users then display site-specific information by clicking on the appropriate spot in the background.

Specifying a custom background image

1. Right-click the background of the window you want to customize and select **Properties** to open the **Window Properties Configuration** dialog.

2. Select the **Image** tab.

3. Select Custom image and click **Browse**.

4. Select the image file you want to use and click **Open**.

5. Click **OK** to set this image as the background.

Query Server: setting a default Query Server for a window

Use the Query Server tab to specify a logged data source for any window in a user diagram. The Query Server that you select provides the default source of logged data for all Event Log Viewers, Data Log Viewers and Grouping Objects within the window.

Setting a default Query Server for a window

1. Right-click the background of the window and select **Properties**.

2. Click the Query Server tab in the Window Properties Configuration box.

3. Select either **Inherit from parent icon** or **Custom** Query Server.

   – Select **Inherit from parent icon** for the active window to default to the same Query Server used by the parent icon. (The parent icon is the diagram object that you clicked to open this window.)

   – Select **Custom** Query Server then click **Edit Link** to open a dialog where you can select a new Query Server link.

   When you click the **Edit Link** button, the **Create Link** dialog opens.

Use the **Create Link** dialog to specify a query:

1. Double-click the Query Server that you want from the Nodes box.

2. Double-click Query Modules from the Managers box.

3. Double-click the Query that you want from the Modules box.

4. Double-click the query register that you want from the Output Registers box.

5. Click **OK** to save your changes.

The name of the selected Query Server and register are displayed as the Group Name on the Query Server tab of the Window Properties Configuration dialog.

Node: Setting the default node for a window

Use the Node tab to specify a real-time source for any window (or grouping object) in a user diagram. The node that you select provides the default source of real-time data for all numeric, status, and control objects within the window. It also is the parent node for any Data Log Viewer or Event Log Viewer objects placed in the window.

Setting a default node for a window

1. Right-click anywhere in the background of the window and select **Properties** from the pop-up menu to open the **Window Properties Configuration** dialog.

2. Select the Node tab.

3. Do one of the following:

   - Select **Inherit from parent icon** to use the same node as the parent icon (the diagram object you double-clicked to open the window).

   - Select **Custom** to select a different node. Select ION and click **Select** to display the **Select Node** dialog. Select a default real-time source for the window from the nodes available.

4. Click **OK** to save your changes.

Action: Specifying an action for a window

You can specify an action for Vista to perform upon opening or closing a particular window. For example, if you want Vista to automatically engage a modem tool (such as the Connection Manager) every time you open a certain grouping window, you can specify the tool as an action.

Specifying an Open or Close action

1. Right-click the background of the user diagram or grouping window and select **Properties** to open the **Window Properties Configuration** dialog.

2. On the **Action** tab, select Command line on open or Command line on close.

3. Click **Browse** to open the **Browse** dialog, locate the program you want, select the .exe file, and click **Open**.

   The specified file and path name appears in the box below the selected option.

4. Click **OK** to save your changes.

NOTE: To remove an action, clear the check box beside the option.

Updates: Setting the time interval

Data that is not updated is considered stale data. Vista detects stale data as it occurs and identifies it with a colored border around the diagram object (the default color is yellow).

On the **Updates** tab, you can specify the interval of time Vista uses to detect stale data. The **Updates** tab has two sections:

- **Update Period** allows you to specify how often data in that window is refreshed.
- **Stale Data** allows you to specify whether data is highlighted and how old data must be to be considered "stale".

NOTE: The Update Period and Stale Data settings also apply to Diagrams.

Changing the Update and Stale Data settings

1. Right-click the background of the window and select **Properties** to open the **Window Properties Configuration** dialog.

2. Select the **Update** tab.

3. Enter a numeric value in seconds in the **Update data period** or **Stale data timeout** fields to set the intervals.

4. Select or clear the **Highlight stale data** check box to enable or disable checking for stale data.

5. Click **OK** to save your changes.

To change the color of the stale data border, close the Window Properties Configuration dialog, then select **Options > Flag Colors** to open the Flag Colors dialog. Click **Edit** beside "Stale Flag Color" to open a dialog where you can select a different color.

Chapter 5: Designer

Use Designer to configure nodes on your network that are based on ION Architecture. The node can be a software component or a hardware device.

> **NOTE**: Only nodes based on ION Architecture can be configured using Designer.

Designer's graphical user interface helps visualize a node's configuration. You can link ION modules together to customize a node for your particular application. You can also cut, copy and paste functionality within a single node or between different nodes.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

Do not use a controlled output for any safety critical application due to possible unexpected changes of state during power cycles, power outages, configuration changes, or firmware upgrades.

Failure to follow these instructions can result in death or serious injury.

Only personnel with a thorough understanding of ION architecture and the system in which the meters and software are installed can configure ION modules and registers.

For more information on ION Architecture, ION modules or ION registers, see the *ION Reference*.

Getting started with Designer

Designer is a programming tool intended **for advanced users only**. Familiarity with ION architecture is essential, as Designer is based on ION architecture.

The functions used in traditional power monitoring are treated as discrete modular objects that can be interconnected and customized. These objects, known as ION modules, are the basic building blocks that comprise a node.

Designer is used to make any changes to the operation of an ION-compliant node. ACCESS devices, the Virtual Processor and Log Inserter are all configured using Designer.

For those already familiar with Designer, see Designer shortcuts for a list of the most commonly used commands.

Starting, logging into and exiting Designer

To start Designer, use one of the methods outlined in Starting a component of WinPM.Net or open Management Console and select **Tools > System > Designer**.

For instructions on logging in to or exiting Designer, see Logging into a component and Ending a session, respectively.

Node diagrams and the Designer interface

This section describes the components of the Designer interface and of node diagrams.
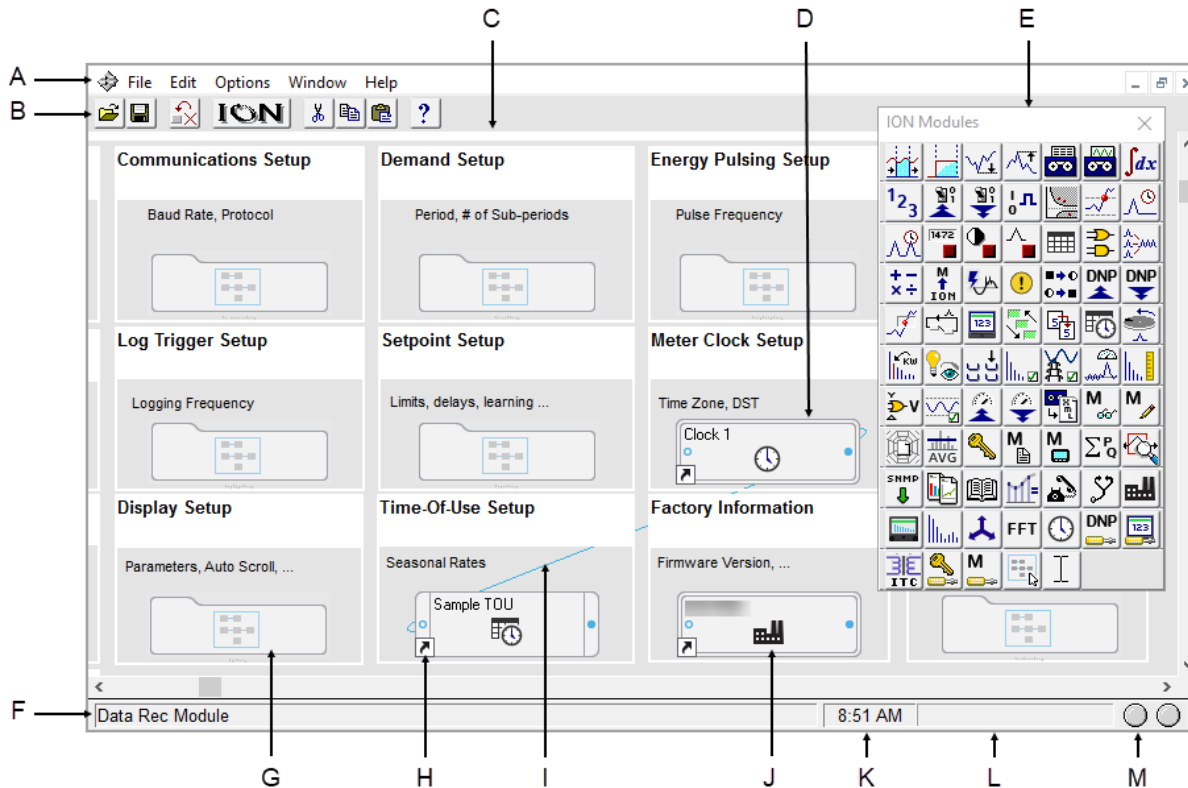
A	Menu bar	B	Toolbar	C	Workspace with open node diagram	D	Core module (indicated by double border)
E	Toolbox	F	Status line	G	Grouping object	H	Shortcut icon
I	Module link	J	Module icon	K	Time display	L	Progress indicator
M	Communication status lights						

The Designer interface

The Designer interface consists of a main screen with a title bar, a menu bar, a toolbar, a workspace, and a status bar. When in Edit mode, the ION Modules toolbox is also displayed.

Title bar

The title bar displays the name of the program, the user name, and the name of the active node diagram.

Menu bar

Below the title bar is the menu bar. Click on a menu name to display a list of available commands.

Toolbar

The Toolbar offers quick access to the commands used most frequently. Each command offered on the toolbar is also available from the menu bar.

A	Open		E	Cut
B	Save		F	Copy
C	Reset		G	Paste
D	Create a link		H	Help

When you point at a toolbar button, a description of it appears in the status bar at the bottom of the screen and on a small ToolTip beside the button.

Workspace

The main area in the center of the screen is the workspace, where you view and edit node diagrams.

Status bar

The status bar displays status information. The components, from left to right, are:

- **Status line**: Describes any active processes and provides brief descriptions of the currently selected command or toolbar button. For example, when you move the pointer over a toolbar button or click on a menu name, a brief description of the item appears in the status line.

- **Time display**: Displays the current time on the workstation.

- **Progress indicator**: Depicts the progress on an action being performed.

- **Communication status lights**: Indicate if Designer is currently receiving and transmitting data on the WinPM.Net network.

The node diagram

A node diagram is a graphical representation of an ION-compliant node. The node diagram displays the actual configuration of the node. Module icons represent the ION modules in the node, while lines drawn between these module icons show links between the ION modules.

In some cases, groups of modules are organized inside grouping windows. When closed, a grouping window appears as a grouping object (an icon that looks like a folder). Click a group object to open the grouping window that contains the module icons.

Using a node diagram

The node is graphically displayed as a node diagram in Designer. The node diagram contains a number of ION modules, which represent the node's current configuration. You can change the configuration of the node simply by editing its node diagram. For example, when you delete a module from a node diagram, the module is removed from both the diagram and the node. You can add, configure, link and delete modules on the node by making the corresponding changes to the module icons in the node diagram. A module's setup registers can be accessed and changed through the module icon.

Use Designer to configure hardware nodes (for example, ACCESS meters) or software nodes (for example, Virtual Processor or Log Inserter).

NOTICE

LOSS OF DATA

Do not leave hardware or software nodes open in Designer, as this stops log downloads from the device.

Failure to follow these instructions can result in permanent loss of data.

Default node diagrams in Designer

When a meter is opened in Designer, the default diagrams for the appropriate meter and default template are loaded automatically.

Opening a node diagram

1. Select **File > Open** to open the **Select Node** dialog.

2. Select the node you want to open then click **OK**.

   When you select a node from the list, Designer communicates with the specified node and opens the applicable node diagram.

NOTICE

UNINTENDED DEVICE OPERATION

Do not open a node diagram if the node is currently being configured from its front panel.

Failure to follow these instructions can result in unplanned configuration changes.

When Designer loads a node diagram, it compares the diagram with the configuration of the node. If there are any discrepancies between the diagram and the node, Designer updates the diagram to match the node:

- If a diagram depicts a module that does not exist on the node, Designer deletes the module icon from the diagram.

- If a node contains a module that is not depicted in the diagram, Designer adds the module icon to the diagram.

- If the links between modules differ from node to diagram, Designer adjusts the diagram to match the node.

A node diagram typically does not differ from the node's true configuration unless the node's configuration was changed through other means (for example, with a remote display unit.)

Saving a node diagram

Saving a node diagram serves two purposes: it saves your changes to the diagram, and applies your changes to the node.

- To save the active node diagram, select **File > Send & Save**.

- To save all open node diagrams, select **File > Send & Save All**.

Closing a node diagram

To close the node diagram(s):

- Select **File > Close** to close the active diagram, or

- Select **File > Close All** to close all open diagrams and windows.

If you attempt to close a diagram without saving your changes, Designer displays a message with a list of the modules affected.

You can then do one of the following:

- Click **OK** to return to the diagram and save your work before closing the diagram.

- Click **Close Anyway** to discard any unsaved changes and close the diagram.

NOTE: If you reopen a diagram containing unsaved changes, Designer may list the unsaved changes as offline modules.

Display mode versus Edit mode

There are two display modes in Designer, which affect how you navigate a node diagram. Whether or not the toolbox is shown indicates the mode you are in: if the toolbox is open, you are in edit mode; if it is closed, you are in display mode.

Display mode allows you to view the node diagram without making changes. To prevent accidentally moving or deleting modules or links, use Display mode when navigating node diagrams. In Display mode, **single-click** grouping objects to open associated grouping windows.

Edit mode allows you to configure the node and the appearance of the node diagram. In Edit mode, **double-click** grouping objects to open associated grouping windows. **Right-click** objects or icons to view configuration options.

For more information on the Toolbox, see Using the ION modules toolbox.

Designer shortcuts

Designer provides several shortcuts to assist in programming the nodes.
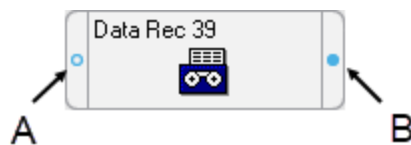
A	Input
B	Output

The following table lists the different mouse and keyboard combinations you can perform on a module's input or output symbols, and their corresponding functions:

Action	Result
Left-click input	The list of inputs appears - you can select an input and link it to another module's output register
SHIFT + left-click input	The list of inputs appears with the current input values displayed in square brackets
Right-click input	The Delete Links dialog appears - you can break links from this dialog.
SHIFT + right-click input	The Delete Links dialog appears, showing inputs and current input values in square brackets
Left-click output	The list of output registers appears
SHIFT + left-click output	The list of output registers appears with the current register value displayed in square brackets
CTRL + left-click output	The list of setup registers appears - these setup registers can be linked to inputs on other modules
SHIFT + CTRL + left-click output	The list of setup registers appears with the current register settings displayed in square brackets
Right-click output	The list of output register owners appears - you can break links from this dialog
CTRL + right-click output	The list of setup register owners appears - you can break links from this dialog

NOTE: To create a shortcut to an ION module, see [Creating a shortcut to an ION module](#).

Designer icons

In each node diagram, Designer uses icons to represent the configuration of the node. There are two basic types of icons: module icons and grouping icons.

Module icons represent ION modules located on the node. Grouping icons represent a group of module icons.

NOTE: A module icon with a symbol in the lower corner is a shortcut.

Module icons

All module icons share four common elements: a label, a graphic, an input symbol and an output symbol.
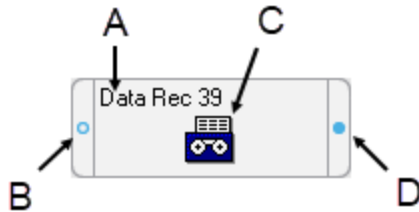
A	Label	C	Graphic
B	Input Symbol	D	Output Symbol

- The **label** displays the name of the module.
- The **graphic** shows a picture to distinguish one type of module from another.
- The **input** and **output** symbols provide access to the module's input and output registers. Click the left symbol to display a list of inputs; click the right symbol to display a list of output registers. See Linking ION modules for more information.

Module icons with a double border represent core modules; module icons with a single border represent standard modules. For more information on core and standard modules, see Core modules and standard modules.

Grouping icons and grouping windows

Grouping icons represent grouping windows. Click a grouping icon (left) to open its associated grouping window (right).
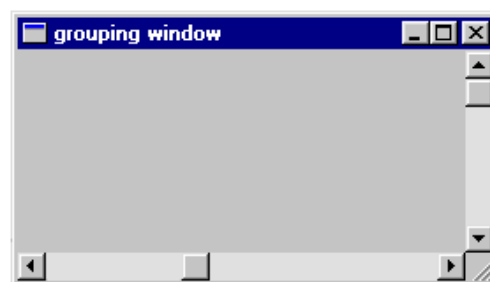
A grouping window acts as a folder or sub-window where you can store icons that you want to keep together. For example, you can use grouping windows to group module icons by application—a single grouping window may contain all module icons required for demand or advanced setup routines.

You can use a grouping window much like a standard window. You can create, rename or delete a grouping window; you can move a grouping window within a diagram, and you can move a module into or out of a grouping window.

When you finish editing the grouping window, close it to minimize the window to its corresponding grouping icon in the node diagram.

Opening a grouping window

Click on the grouping icon.

Closing a grouping window

Click the **Close** button .

Moving icons into a grouping window

To move an icon (or group of icons) to or from an open grouping window, select it and drag it to
the new location.

1. Click on a grouping icon to open the grouping window.

   Drag the grouping window by the title bar and position it so that you can see the icon(s) that
   you want to move.

2. Select the icon(s) you want to move and drag the selection into the grouping window.

When you move a linked module icon into a grouping window, the module remains linked but any
lines that represent links to modules outside the window disappear.

Creating a new grouping window

To create a new grouping window, drag the grouping object from the toolbox onto the node
diagram.

The ION modules toolbox is a collection of ION modules and generic tools that you can add to a
node diagram. If the ION modules toolbox is not displayed, select **Options > Show Toolbox**. See
Using the ION modules toolbox for more information.

When you point to an object in the toolbox, the name of the object is displayed on a ToolTip. You
can use ToolTips to identify the grouping object in the toolbox.

Renaming a grouping window

When you create a new grouping window, the default name is that of the node diagram. To
change the name of a grouping window, rename the grouping icon:

1. Right-click the grouping icon to open the **Grouping Object Configuration** dialog.

2. Select **Use Custom** from the Caption section and type the new name into the text box.

3. Click **OK**.

Changing the font of a grouping window

To change the font, font style and size of a caption:

1. Right-click the grouping icon to open the **Grouping Object Configuration** dialog.

2. In the Font section, do one of the following:

   - To use the font of the parent window, select **Inherit from parent window**.

   - To use a different font, select **Custom**, then click **Font** to open the **Font** dialog. Select
     the font options you want then click **OK** to continue.

3. Click **OK**.

Changing the position of the caption

To change the location of the caption:

1. Right-click the grouping icon to open the **Grouping Object Configuration** dialog.

2. Select either **Top** or **Bottom** in the Position section to specify a caption position above or below the grouping icon.

3. Click **OK**.

Deleting a grouping window

> **NOTE**: When you delete a grouping window, any modules inside are deleted as well. Designer displays a message before deleting modules. See Deleting or cutting an ION module for more information on deleting modules.

1. Select the grouping icon of the window you want to delete.

2. Press the **Delete** key. If the grouping window contains modules, Designer displays a list of modules that will be deleted.

Displaying default or custom icon labels

Module icons can display two types of labels: Default (left) and Custom (right).

- **Default labels**: All modules have a default label. The default label identifies the module by type and, if applicable, by number. It appears in node diagrams, user diagrams and event logs unless a custom label is available.

- **Custom labels**: Custom labels are available only if they have been preconfigured by a user. A custom label usually identifies a module by its function or purpose. Custom labels are useful for describing modules, setup registers, output registers, and Boolean ON/OFF conditions. For information on adding custom labels, see Customizing a module label.

Displaying default labels or custom labels

Select **Options > Show Default Labels** to toggle the option on or off.

- When Show Default Labels is selected, default labels are displayed.

- When Show Default Labels is not selected, custom labels are displayed.

This is a global setting that applies to all node diagrams.

Moving icons in a node diagram

You can move module and grouping icons anywhere in a node diagram. This is useful for organizing your diagram (for example, by function or type) so that you can easily find a particular module or visualize the workings of your node's configuration. You can move a single icon or group of icons within a single node diagram, or into an open grouping window.

> **NOTE**: Moving the icons in a node diagram does not affect the function of the modules or the node in any way.

Moving icons

1. Select the icon(s) you want to move.

   When a grouping icon is selected, all icons within its grouping window are automatically selected as well.

2. Drag the selected icon(s) to the new location.

NOTE: When selecting a group of icons, you can only choose icons from within the active window.

Arranging icons in a node diagram

You can use the Align and Grid options in the Layout dialog to adjust the spacing and layout of the module and grouping icons in your node diagram. Use the Align option to specify the horizontal and vertical arrangement of the selected icons and the Grid option to activate and configure invisible guidelines.

NOTE: Arranging the icons in a node diagram does not affect the function of the modules or the node in any way.

Changing the order of layered or stacked icons

If an icon appears on top of another icon that you want to bring to the front, select the icon that is sitting on top, then click CTRL+K (or click **Edit > Send to Back**).

Aligning icons

The **Align** tab in the Layout dialog allows you to specify the horizontal and vertical alignment of selected objects.

To align objects:

1. Select the icons that you want to align.

2. Select **Edit > Layout** to open the **Layout** dialog.

3. Select the **Align** tab and set the horizontal and vertical alignment of the icons.

   The options in each area determine the criteria by which you can align icons. For example, if you select **Left sides** under Horizontal and **Space evenly** under Vertical, Designer aligns all objects in the selection by their left sides (using the leftmost object for reference) and distributes them evenly along a vertical axis.

4. Click **OK**.

Using the grid

The **Grid** tab in the Layout dialog activates and configures an invisible grid. Icons placed onto the grid snap to the nearest gridline to help align and space them evenly.

To activate or deactivate the grid:

1. Select **Edit > Layout** to open the **Layout** dialog.

2. Select the **Grid** tab.

3. Select one of the following options:

    –   **No Grid** turns the grid off.

    –   **Grid size in pixels** activates the grid. Type a number in the Grid size in pixels box to specify the size of the grid (distance between grid lines).

4. Click **OK** to save your changes.

Designer windows

To create links between several node diagrams or grouping windows, you need to position the open windows so you can see all the modules involved. Use the Window menu to organize open windows in the workspace.

Arranging windows in your workspace

To arrange windows, do one of the following:

- To locate an open window and move it to the foreground, select the Window menu and select the window name from the list of open windows.

- To arrange all open windows in the workspace, select **Window > Arrange All**.

Changing a window's background color

By default, the background color of all windows in a node diagram is light gray. To select a new background color for a window:

1. Right-click the background of the window and select **Background Color** to open the **Color** dialog.

2. Select the color you want or click **Define Custom Colors** for more options.

3. Click **OK**.

When choosing a background color, consider how the color affects the visibility of icons and links. Links, highlights and shadow effects used to depict the various states of an ION module may not appear against certain background colors. For this reason, you should avoid using certain colors for your background, particularly white, dark gray and black.

Changing a window's default font

You can specify a default font to be used for all module icons, grouping icons and text boxes in a window.

1. Right-click the background of the window and select **Default Font** to open the **Font** dialog.

2. Specify a new default font, size and style.

3. Click **OK**.

Fonts can also be specified individually for grouping windows and text boxes. See Changing the font of a grouping window and Changing the font of a Text Box for more information.

ION module overview

After you have opened a node diagram, you can view the node's configuration or edit it by working with its ION modules. As described earlier, each module performs a specific function. Using Designer, you can add, remove or link modules together to customize the function of the node.

This section explains how to add and remove ION modules from your node diagram. It describes the basic types of modules, and explains how you can organize your node diagram by grouping modules together, arranging them, and labeling them.

For more information on ION modules and architecture, refer to the ION Reference.

Working with ION modules

The following sections outline some considerations to remember when working with ION modules.

Output and setup registers

A register is a type of memory bank that stores a module's linking and configuration information. Each module uses two types of registers: output registers and setup registers.

Module status: online and offline

The terms "online" and "offline" describe whether a module is currently active or not. A module is described as online when it is functioning normally (monitoring its input and updating its output registers). An offline module is inactive.

Anytime you send and save changes to a node, Designer temporarily takes the affected modules offline and programs them with your changes. After they have been programmed, the modules are then placed back online.

Normally, this is a routine procedure. However, certain circumstances may cause a module to remain offline. For example, if the node lacks sufficient processing power to operate the module, the module remains offline.

You can easily identify any offline modules: all offline modules are highlighted with a red outline. In addition, whenever you open a node diagram, Designer displays a list of all offline modules in the node.

> **NOTE**: If you have offline modules in your node, you may be able to put them back online by selecting **File > Send & Save**.

Module status: programmed vs. pending

The terms "programmed" (left) and "pending" (right) refer to whether or not a module's representation in the node diagram matches the node itself.

Programmed: A programmed module is one that is the same both in the node diagram and in the node. The links, labels and setup register values in the node diagram exactly match those on the node. When you first open a node diagram, all the modules appear programmed since they are read directly from the node itself and then displayed in the node diagram.

Pending: A pending module is one where changes have been made to the module in the node diagram and not yet sent to the node, so the node diagram does not match the node. The borders of pending module icons appear as a dashed line.

If you change a module in the node diagram (for example, re-link it or edit its setup registers), it does not match the node until you select **File > Send & Save**. Until it is saved, the edited module is depicted as pending to indicate that it does not match the configuration on the node. When you send and save your changes to the node, the module becomes programmed.

Core modules and standard modules

There are two basic types of ION modules: core (left) and standard (right).

Core modules are required ION modules that are fundamental to the operation of the node. You cannot add or delete core modules and, in some cases, you cannot configure them. Core module icons are identified by a double border around the edge. The Power Meter module is an example of a core module.

Standard modules are reusable ION modules. Generally, most modules in a node are standard modules. You can add or delete standard modules as required. Standard modules icons are identified by a single border. The Setpoint module is an example of a standard module.

Persistent modules

Similar to core modules in that they cannot be added or deleted, persistent modules are standard modules that have been converted to core modules. These modules are created at the factory, and can never be removed from the meter frameworks. Persistent modules are represented by icons with single borders.

An example of a persistent module is External Pulse module #1 on the ION8600 meter. This module pulses when the Demand Reset switch is pressed on the meter.

Using ION modules

You can change the function of a node by editing the appropriate module(s) in the node diagram and linking modules together.

This section describes how to locate an existing module, how to add a new module, and how to delete a module. It also explains how to create a shortcut to a module and how to view the contents of a module in text format.

After you have located or added the module you want, you can configure it using the procedure described in Linking ION modules.

Using the ION modules toolbox

The ION Modules toolbox is a collection of ION modules and generic tools that you can add to a node diagram. To display the ION Modules toolbox, select **Options > Show Toolbox**.
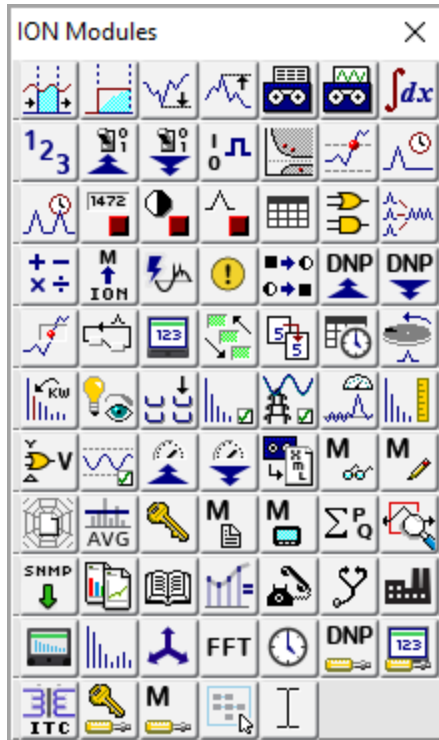
The type and quantity of modules offered on the toolbox varies depending on the type of node you are configuring; however, the generic tools (the grouping and text objects) are available for all nodes.

NOTE: To identify a module in the toolbox, point to it with the mouse—the module type is displayed as a ToolTip and on the status line at the bottom of the screen.

To add a module to your node diagram (and to the node itself), drag its icon from the toolbox into the diagram window.

Opening the ION modules toolbox

To open the toolbox, select **Options > Show Toolbox**.

Closing the ION Modules Toolbox

Click the **Close** button ✕ or clear **Options > Show Toolbox**.

Locating an existing ION module

Before you can edit a module, you must first locate it in the node diagram. To find a module in a node diagram, you can use the module tray feature of the toolbox.

Locating an existing module from the Module Tray

1. To display the module tray, right-click the toolbox icon for the type of module you want to locate. The module tray appears beside the toolbox.
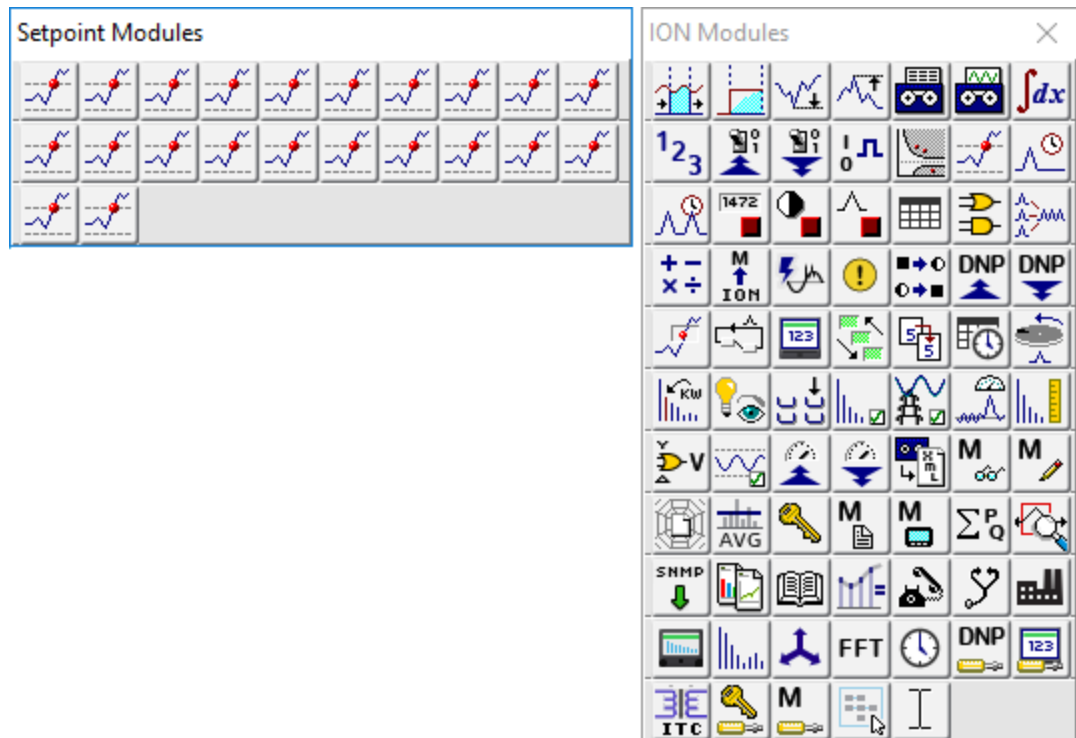
The module tray displays links to all modules of that type that are currently in use on the node—for example, in the image above, there are 22 Setpoint modules in use by the node. To identify a link in the tray, point to it—the module's label is displayed on the status line at the bottom of the screen.

2. To locate a particular module, click its link in the module tray. Designer locates the module and displays it in the center of the workspace. If the module is located inside a grouping window, Designer opens the window to display the module.

TIP: To close the module tray, click on any module icon in the toolbox.

Adding a new ION module

Most nodes are pre-programmed at the factory to provide the most commonly required functions. However, if the factory default configuration is not appropriate for your application, you can add a new module.

Adding a module on a node

Drag the icon of the module you want to add from the toolbox onto the node diagram.

As you drag the icon, the cursor changes shape to indicate you are placing an object.

Designer adds the module and places a pending module icon into the diagram. You can then link and configure the module.

See Linking ION modules for more information on programming modules.

Deleting or cutting an ION module

To remove a standard module from a node, delete its icon from the node diagram. By removing unnecessary modules from the node, you can free up resources for use elsewhere.

NOTICE

UNINTENDED DEVICE OPERATION

Do not delete any ION modules without knowing which modules are dependent on it.

Failure to follow these instructions can result in the unintended operation of the remaining modules.

NOTE: Each time a module is deleted, all of its output registers, setup registers and input links are deleted as well. Removing this information from the node affects the operation of any modules that depend on it. For this reason, you should be aware of all dependent modules (also known as owners) before you delete anything.

Checking for dependent modules

When two linked modules are located in the same window, Designer depicts the link as a line that runs from the output register of one module to the input of another. You can easily identify the dependent module (or owner) by the line running to its input. However, if one of the linked modules is located in a different window, the connection is not visible. A more effective way to locate a module's owners is to view the owners of its registers.

Viewing the owners of a module's setup and output registers

You can view a register's owners to quickly identify all dependent modules on the node. You can determine what purpose the modules serve, and then decide whether to delete them or not.

1. Do one of the following to open the **Owners** dialog:

    – To view the owners of a module's output registers, right-click the output symbol.

    – To view the owners of a module's setup registers, hold the CTRL key and then right-click the output symbol.

    The dialog lists all of the module's registers and any dependent modules on the node (dependent modules on other nodes are not displayed).

2. Click a register in the Registers list to display the owners of that register in the Owners list on the right. Owner information includes the module name or label and its relevant input.

3. If necessary, delete the link between a register and one of its owners. Select the owner from the Owners list and click **Unlink**. (See Deleting links for more information on deleting module links.)

4. When you have finished viewing the register owners, click **OK**.

Viewing owners on other nodes

Displaying a module's register owners is an effective way to locate dependent modules on the selected node but it does not show any dependent modules located on another node.

If you remove a module with a dependent module on another node, the dependent module's inputs are not reset. It continues to look for the deleted output register.

If you are not sure whether a module has dependents on other nodes, check with the person who programmed them to ensure you do not disrupt the operation of any modules on that node. If you still want to remove the module, you may want to directly delete the link on the other node first. In this way the module is not inadvertently linked to the next module to occupy the location of the deleted module.

Deleting or cutting an ION module

After you have determined that you can remove an ION module without disrupting other functions, there are two ways to remove it: delete and discard the module or cut the module to paste it in another location.

In either case, Designer immediately removes the icon from the node diagram and deletes the module from the node. The difference is, when you delete a module, it is discarded; when you cut a module, it can be restored (by selecting **Edit > Paste**).

NOTE: You do not have to select **Send & Save Changes** to delete a module. Designer immediately removes the module from both the diagram and the node.

Removing an ION module from the node

1. Select the icon(s) of the module(s) you want to remove.

2. Do one of the following:

   - To cut the module(s) to paste it in another location, select **Edit > Cut**.

   - To discard the module(s), press the **Delete** key.

NOTICE

UNINTENDED DEVICE OPERATION

Do not delete any ION modules without knowing which modules are dependent on it.

Failure to follow these instructions can result in the unintended operation of the remaining modules.

NOTE: If a module has a dependent module on another node, Designer does NOT alert you of the dependency when you delete it. Before you delete a module, ensure that you are aware of all links to modules on other nodes.

Designer displays a summary of the proposed changes.

The summary lists all selected modules, and identifies those in the selection that will be deleted (including those with dependents on that node) and those that will not. Select a module in this list to display any additional information available in the field at the bottom of the dialog.

3. Click **OK** to remove the selected modules (or **Cancel** to abort the procedure).

Designer removes the module icon from the node diagram and deletes the module from the node itself.

> **NOTE**: If you delete a shortcut icon, the original module is not affected. However, when you delete the original module, all shortcuts to that module are also deleted.

Viewing a node or module as text

You can use the View Diagram Text command to display as text the complete contents of one or more modules or of an entire node. Use this option to view the specified information on your screen so you can sort it, perform analysis, create a detailed record of your node's configuration, or print a copy for your records.

1. Select the module(s) that you want to display.

> **TIP**: To select all modules in the node, left-click on the background of the main node window, being careful not to click on a module icon, then select **Edit > Select All**.

2. Select **Options > View Diagram Text** to open the **Text View** screen. The text of any modules that are offline appears in red.

- The Text View screen offers several ways to view, sort and find information:

- In the View dropdown list, select one of two view modes:

   – Use **Archive** to display information useful for archiving the node's configuration.

   – Use **Diagnostic** to display diagnostic information on the selected module(s).

- In the **Sort by** dropdown list, select the criteria by which you want to sort the data:

Archive View	Diagnostic View	Sorts by
Default	Default	Module label
Module Class	Module Class	Type of module
Module Name	Module Name	Default label
	Module State	Displays offline modules first followed by online modules. Sorts alphabetically within each group.
	Module Update Period	Displays event driven modules first, then inactive (not updating) modules, followed by high-speed (one cycle updates) modules, and finally high-accuracy (one second updates) modules. Sorts alphabetically within each group.

- In the **Find** text field, type a text string and click **Find** to find that text string in the data.

To reset the text view after you have made a configuration change, click Refresh. If you want to print a copy of the text view information, click Print. When you are finished viewing the text view information, click Close.

Creating a shortcut to an ION module

To save time switching between modules in different windows, create a shortcut icon in one window that represents the original module in the other window.

A shortcut icon is identified by a symbol in the lower left corner of a module icon.

Creating a Shortcut Icon

Hold down SHIFT+CTRL then drag the module icon(s) to the window where you want to place the shortcut(s).

After you have created a shortcut icon, you can use it in the same way you use the original module icon. Both icons support the same features. However, if you delete the shortcut icon, the original module icon is not deleted; whereas if you delete the module icon, it deletes all shortcuts to that module.

> **NOTE**: You cannot create more than one shortcut icon per window for a single module.

Adding a text box to a node diagram

You can place a text box anywhere in a node diagram, and move it or resize it as necessary. You can use a text box to describe any process or function in your system; for example, you can label individual components of your system or display a block of descriptive text or instructions.

Adding a Text Box

1. Drag a **Text Box** object () from the toolbox onto the node diagram.

   A text box opens in the diagram with the default message: "Your text goes here."

2. Right-click the text box to replace the default text with your own text. The **Text Box Configuration** dialog opens.

   – Select the **Edit Text** tab, then do one of the following:

   – To display the name of the node in the text box, select **Use Default**.

3. To display your own message, select **Use Custom** and type your text into the field provided.

4. Click **OK**. The text box displays your specified text.

Resizing a Text Box

To resize a text box:

1. Select the text box.

2. Do one of the following:

   – To make the object larger or smaller, drag a corner handle.

   – To stretch the object horizontally or vertically, drag a middle handle.

Changing the font of a Text Box

The font, size and style of the text used in the text box is based on the default setting specified for the active window. If you want to use a different font, you can specify a new font for the text box:

1. Right-click the text box to open the **Text Box Configuration** dialog.

2. Select the **Text** tab.

3. In the Font section, do one of the following.

   – To use the font specified for the parent window, select **Inherit from parent window**.

   – To use a different font, select **Custom**, then click **Font** to open the **Font** dialog. Specify a font, style, and size. Click **OK** to continue.

4. Click **OK**.

Changing the position of the Text

To change the position of the text in the text box:

1. Right-click the text box to open the **Text Box Configuration** dialog.

2. Select the **Text** tab.

3. In the Position section, select **Left**, **Center**, or **Right**.

4. Click **OK**.

Adding a border or background color to a Text Box

To add a border to a text box or change its background color:

1. Right-click the text box to open the **Text Box Configuration** dialog.

2. Select the **Box** tab.

3. Do one or more of the following:

    –   To add a border, select **Show** in the Border section and click **Color** to select a border color.

    –   To set the width of the border, type a border width in the "Width in pixels" field in the Border section.

    –   To add a background color, select **Custom** in the Background Color section and click **Color** to select a background color.

4. Click **OK**.

Linking ION modules

Linking ION modules together is the foundation of programming ION-compliant nodes in WinPM.Net. Although an ION module can perform a function in isolation, the strength and flexibility of the ION architecture comes from the ability to interconnect modules to build more sophisticated functions.

Choosing which ION modules to link

The first step in linking modules is to decide which modules you want to use. When choosing a module to link, you have two choices: you can add and link a new module or you can re-link an existing module.

In some cases, you may not want to add a new module. For example, if all the modules of a particular type are already used, you need to re-link one of them to perform the new function. Most devices are preconfigured at the factory to provide common measurements and calculations. If any of these factory defaults are unnecessary in your application, you can unlink the modules involved and reuse them.

Plan ahead before you add new modules. Unlinked modules are wasted resources.

You may find a linked module that you can use to add a new link, in order to augment the function it is performing. For example, if you have a Setpoint module that triggers a waveform recording, then later decide to use this trigger to reset a counter, you can link that Counter module to the same Setpoint that controls the Waveform Recorder module.

Linking ION modules graphically

Linking modules graphically means pointing and clicking a module's input, then clicking another module's output register to link it (note that both input and output register classes must be compatible). You can use this method to link to modules on the same node or to create links across nodes.

You can use the autolinking feature to speed up the process of linking ION modules. Typically, linking ION modules involves connecting the output register of one module to the input of another. In many cases, only one of the module's inputs have the same register class as the selected output register.

If autolinking is enabled, Designer automatically selects the appropriate input and establishes the link. If necessary, it overwrites an existing link. This relieves you from having to manually select the input; however, you cannot see what input you are linking to.

By default, autolinking is disabled. You should only use autolinking if you are thoroughly familiar with all the inputs and output registers of the modules you are working with so that you do not inadvertently create undesirable links.

Enabling autolinking

To enable autolinking, select **Options > Enable Autolinking**. A check mark beside the option indicates it is selected.

Disabling autolinking

To disable autolinking, clear **Options > Enable Autolinking**.

Linking ION modules on the same node

If the selected node is a meter such as the 9610, the process of configuring the node primarily involves creating links between modules on that node.

NOTE: To cancel a link-in-progress, click anywhere in the background of the window or press the ESC key.

Linking modules using the point and click method

1. To list a module's output registers, click the output symbol on the right side of the module icon. To list the module's setup registers, hold the CTRL key while clicking the output symbol.

   Either a menu opens or, if the module has many registers, a dialog opens listing the available registers.

2. Select the register you want to link:

   – In the menu, click the register you want to select.

   – In the dialog, double-click the register (or click it then click **Select**).

3. Move the cursor towards the module you want to link to. The cursor changes and a dotted line follows it across the node diagram. This indicates you are in the process of creating a link and it shows where the connecting line will appear in the node diagram.

If you link to a module that is in a different window than the original module (either in a different node diagram or grouping window) the dotted line disappears but the cursor still indicates that a link is in progress.

If the module you want is obscured by another window, click on the title bar of the window containing the module. This brings the window to the foreground without canceling the link. (If you click on the background of the window, the link is cancelled.)

4. Click the input symbol of the module that you want to link to display the module's inputs. Either a menu opens or, if the module has many registers, a dialog opens listing the module's inputs.

    If the input is already linked, the label of the register it is linked to is displayed beside the input. If you select the input, the existing link is overwritten.

    The inputs that are of a different class than the selected output register are grayed out to indicate you cannot select them.

5. Select the input you want to link to:

    – In the menu, click the register you want to select.

    – In the dialog, double-click the register (or click it then click Select).

6. Select **File > Send & Save** to save your changes to the node.

The procedure described above can also be performed in reverse order. You can select a module's input first and then link it to another module's output register.

Linking modules with variable number of inputs

Some modules, such as AND/OR modules, Data Recorder modules, and Log Acquisition modules have a variable number of inputs. In these cases, when you click on the input symbol, the menu offers one instance of the variable input as well as any fixed inputs the module may have. When linking to one of these modules, you can select the **NEW Source** item to add a new link or you can select an existing link and overwrite it.

If the two modules are in the same window, the line remains on the screen to show the link between the modules—when you save this change to the node, the line changes from a dotted line to a thin black line to indicate that the link is now programmed on the node.

Linking ION modules across different nodes

If the selected node is a software node, such as the Virtual Processor or Log Inserter, many of the links you create are to modules on other nodes. When you create links across different nodes, it is important to keep track of these links and note what modules are dependent on others. This helps you determine what will happen if you ever want to delete a module that is linked to a module on another node.

NOTE: You cannot link modules on one device to another. For example, you cannot create a link between a module on an 9410 to a module on a different 9410 or to a module on another meter.

1. Open the node diagrams that contain the modules you want to link.

2. Position the diagrams within the workspace so you can see the two modules you want to link.

3. In the first node diagram, click on the output symbol of the module icon. A menu or dialog opens listing the module's output registers.

4. In the list of output registers, click the register you want. (In the case of the dialog, click a register and then click **Select**.)

5. Drag the cursor from the first node diagram to the module you want to link to in the second node diagram. The dotted line ceases to follow the cursor if it crosses the window boundary but the cursor changes to indicate a link is in progress.

6. Click on the input symbol of the module icon in the second node diagram. A menu or dialog opens listing the module's inputs. Those that are of a different class than the selected output register are grayed out to indicate you cannot select them.

7. Click the input you want, or in the case of the dialog, click the input then click **Select**. Designer does not graphically represent links between modules on different nodes so you will not see a line connecting the modules.

8. Select **File > Send & Save** to save your changes to the node.

This procedure can also be performed in reverse order. You can select a module's input first and then link it to another module's output register.

> **NOTE**: If the input is already linked, the label of the register it is linked to is displayed beside the input. If you select this input, the existing link is overwritten.

A note about performance

While creating links, you may notice that the circle at the end of the arrow cursor is sometimes black ( ).

This indicates that Designer is in the process of retrieving information about a module. You can still click on the inputs or outputs of other modules, but Designer aborts the original request. To minimize the amount of communications between Designer and the connected nodes, avoid unnecessary clicking on the input and output symbols.

If you click on several items in a row, you only see the menu for the last item you click; however, Designer initiates many requests for information, which may impact its performance.

Linking ION modules using the ION tree

In some cases, you may want to use the ION tree to select the register for linking. The ION tree is a hierarchical listing of every node, manager, module, and register in the WinPM.Net network. It allows you to select a register on a node without opening the node diagram. Generally, you use the ION tree to create links across nodes.

The ION tree is also the only way to select registers from devices that are not ION-compliant (and use translators to make data available) such as PAC series meters. For example, if you are programming a Virtual Processor node and you want to sum data from several PAC3200s, you must access the PAC3200 register via the ION tree.

Using the ION tree is faster than opening a node diagram. It is useful if you want to link to a register on a node whose diagram has not yet been opened. It is also useful if you want to link to a register, but you are not acquainted with the configuration of the node in which it resides. You can

go straight to the register without having to navigate through an unfamiliar node diagram. Also, since a diagram can only be accessed by one user at a time, you can use the ION tree to link to a register on another node where the diagram is already open.

Linking modules using the ION tree

1. Click the input symbol of the module icon. A list of the module's inputs is displayed.

2. In the list of inputs, select the input you want to link.

3. Click the **ION** button in the toolbar. (To link a module's setup register, hold the CTRL key while clicking this button.)

    The **Create Link** dialog opens, providing access to the ION tree.

4. Specify a node, manager, module and output register: Double-click the node you want in the Nodes box. All managers within the selected node appear in the Managers box. You can then double-click the manager you want, and repeat this procedure for the modules and registers.

5. Click **OK**. The module whose inputs changed appears grayed-out to indicate it is now pending. If both modules are in the same window of the same node diagram, a line appears to represent the link.

6. Select **File > Send & Save**. The new link is established and the module appears programmed.

The procedure described above can also be performed in the reverse order. You can click the **ION** button first, select an output register, and then click the input symbol of a module icon to select one of its inputs.

Deleting links

If you want to disassemble a group of linked modules and use them for another function, you can delete the links between the modules rather than deleting the modules. Designer provides several ways to delete links.

Fixed links

Most links between standard modules can be deleted. However, some links between core modules are fixed and cannot be deleted. These fixed links include:

- Data Acquisition module and all Power Meter modules.

- Data Acquisition module and all FFT modules.

- FFT module and all Harmonics Analyzer modules.

- FFT module and all Symmetrical Components modules.

If you attempt to delete a fixed link, Designer informs you that the link cannot be deleted.

Deleting visible links

If the linked modules are in the same window, simply click on the link line and press the **Delete** key. This works well as long as you know which output register and input are involved. For example, in the image below, to delete the link click on the line joining the Power Meter and the Sliding Window Demand module (its color reverses to indicate it is selected) and press **Delete**.

link line
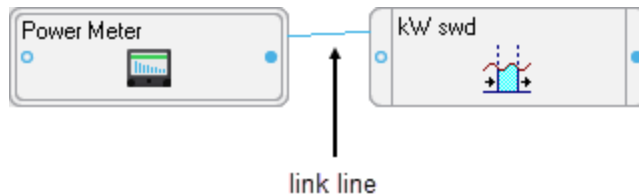
If there are multiple lines between the same modules and you only want to delete one of the links, you may want to use the method described in [Deleting links that are not visible](#) below.

Deleting links that are not visible

If you cannot see a line that links the modules (for example, if they are in different windows), or you need to know which output registers and which inputs are involved, you must use a dialog to delete the link between modules. You can look either at the output register owners of the first module or you can look at the inputs of the second module to see what they are linked to, and then delete the link from there.

Deleting a module's input links

Deleting links at a module's inputs is generally faster than deleting them at the module's output registers because Designer does not need to check the entire node. On the other hand, there is less information available if you view input links. All you see is the label of the output register that is linked to the input. If this is sufficient information, proceed with this method; if not, go to the output registers to delete the link.

Viewing a module's input links

1. Right-click the input symbol of the module icon to open the **Delete Links** dialog.

2. Each input and the label of the output register it is linked to are displayed in the dialog. To delete one or more of these links, select the input and click **Unlink**. The output register label is replaced with dashed lines.

3. When you have unlinked all the registers you need to, click **OK**.

4. Select **Send & Save** to unlink the modules on the node.

Deleting a module's output links

You can also delete links from the Register Owners dialog. This method gives you the most information about the link you are deleting (the output register label and the input and label of the module it is linked to).

Viewing a module's output links

1. Right-click the output symbol side of the module icon to open the **Register Owners** dialog. (To list the module's setup registers, hold the CTRL key while right-clicking on the symbol.)

2. Select the output register you want from the Registers section. The Owners section lists all the module inputs to which the selected register is linked.

3. To delete one or more of these links, select the module input in the Owners list box and click **Unlink**. The input name is replaced with a dashed line.

4. When you have unlinked all of the module inputs you want, click **OK**.

5. Select **Send & Save** to unlink the module inputs on the node.

> **NOTE**: Owners that reside on a different node are NOT displayed in this dialog. If you need to delete a link between modules on different nodes, use the procedure described in [Deleting a module's input links](#).

Configuring ION modules

After you have placed a module in a node diagram, you can configure the various setup parameters of the module. It is not necessary to do this before you link the modules but you should ensure the module is set up correctly before sending the configuration to the node.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

Do not use a controlled output for any safety critical application due to possible unexpected changes of state during power cycles, power outages, configuration changes, or firmware upgrades.

Failure to follow these instructions can result in death or serious injury.

Only personnel with a thorough understanding of ION architecture and the system in which the meters and software are installed can configure ION modules and registers.

Module setup parameters include the module's label; its setup registers, labels and values; and its output register labels. To access these, right-click the module icon.
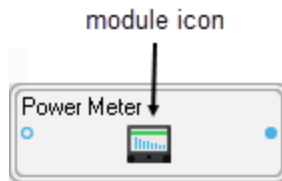
The ION Module Setup dialog opens.

The Setup Registers box lists all the setup registers of the module along with their current values or settings. To change any of the settings or assign a custom label to a setup register, select the register, then click **Modify** (or double-click on the register) to open the **Modify Register** dialog where you can make modifications.

In most cases, configuring a setup register is a simple matter of entering a number, entering text, or choosing an option from a list. (To determine the options or ranges available for a setup register, refer to the module's description in the *ION Reference*.)

When you have completed all your modifications to the module, click **OK** to return to the node diagram. The module icon appears pending until you send and save your changes.

Modifying enumerated, numeric bounded, and string setup registers

If the register you have selected is an enumerated, numeric bounded or string register, one of the following dialogs appear when you click **Modify**:

Enumerated registers

To modify an enumerated register, select an option from the Options list.

Numeric bounded registers

To modify a numeric bounded register, type a number in the Value box. The allowable range is shown under the Value box; you are alerted if you type a number outside this range.

To enter a formatted numeric value, click **Format**. In the **Time Interval Value** dialog, you can enter the numeric value as a date (offset from Jan 1, 1970) or a time interval.

String registers

To modify a string register, type your text in the String box.

Note that when you click **Modify** for a string register that is the connection string for a Log Acquisition module or Query module, the **Modify Database Connection Properties** dialog opens.

Modifying calendar setup registers

The ION Scheduler module has a special class of setup register called a calendar register. Configuring a calendar register is only necessary for nodes that have a Scheduler module. For detailed information on how to configure a calendar register, see the description of the Scheduler module in the *ION Reference*.

Modifying address setup registers

All Distributed Numeric, Distributed Boolean, and Distributed Pulse modules contain an address setup register that references another register. You can specify a new value or clear an existing value as required. (Currently, the Virtual Processor is the only node offering these types of modules.)

NOTE: Currently, you can only link the address setup register to External Numeric, External Boolean, and External Pulse registers.

Modifying an address setup register

1. Right-click a Distributed Numeric, Distributed Boolean or Distributed Pulse to open the **ION Module Setup** dialog.

2. In the Setup Registers section, select a register then click **Modify** to open the **Modify Address Register** dialog.

3. In the **Modify Address Register** dialog, double-click on the node, manager, module and output register you want to use. Click **OK** to continue.

4. Click **OK** in the **ION Module Setup** dialog.

NOTE: You can clear the value of an address setup register. On the **Modify Address Register** dialog, select **Reset Register Value** and then click **OK**.

Parent module and shared setup registers

Setup registers that are surrounded by asterisks (for example, "**PM1 PT Sec**") indicate that the module shares its setup registers with another ("parent") module. An example of this is the High-Speed Power Meter or Meter Units Power Meter module ("HS Power Meter", "MU Power Meter").

They share their setup registers with the parent Power Meter module.

If you try to modify a shared setup register, a prompt identifies the register as shared and gives the name of the parent module.

Navigate to the parent module and change the setup register there.

Using custom labels

You can assign custom labels to modules, setup registers, output registers, and Boolean ON/OFF conditions. All modules and registers have a default label and this label is what you see in node diagrams, user diagrams and event logs unless you specify a custom label.

Creating custom labels allows you to clearly describe what the module and its registers have been set up to do.

However, carefully consider which modules and registers should be assigned custom labels. There is a fixed number of labels available, but there are thousands of registers you can assign them to. Many of these registers do not benefit from custom labels. For example, most setup registers can be left at their default names since the only place their names might appear is in the event log.

NOTE: Some devices, such as the 9300 and 9700, have a 15-character limit for labels.

Customizing a module label

The ION Module Setup dialog allows you to edit the module label. The module label is the text that appears across the top of the module icon in the node diagram. It is also the label that is used in event logs and in the dialog for the ION tree.

1. In the **ION Module Setup** dialog, do one of the following:
   - To create a custom label, select **Use Custom Label** and then type a name for the label in the box.
   - To use the default label, select **Use Default Label**.
2. Click **OK** to return to the node diagram.

Customizing an output register label

By providing a custom label for an output register, you can clearly indicate the information that register contains. For example, if you have linked the Vlna output register of a Power Meter module to a Thermal Demand module, and in turn linked the ThrmDemand output register to a Maximum module, you may want to label the Maximum module's output register as "Va TD Maximum" (instead of using the default name of "Maximum 1").

1. In the **ION Module Setup** dialog, select the output register in the Output Registers area and then click **Modify** to open the **Modify Label** dialog.
2. Do one of the following:
   - To create a custom label, select **Use Custom Label** and then type a name for the label in the box.
   - To use the default label, select **Use Default Label**.
3. Click **OK** to return to the **ION Module Setup** dialog.

Customizing Boolean ON/OFF labels

If the output registers of the module are Boolean, you can create custom labels to associate with the ON and OFF conditions of these registers. These labels can clarify exactly what the ON or OFF conditions represent. For example, if a Boolean output register is used to switch a fan on and off, you could create a Boolean ON label called "Fan Running" and an OFF label called "Fan Not Running."

1. In the **ION Module Setup** dialog, select the Boolean output register and then click **Modify** to open the **Modify Boolean Register** dialog.

2. Do one of the following:
   - To create custom labels, select **Use Custom Labels** and then type names for the Register Label, ON Label and OFF Label in the appropriate fields.
   - To use the default labels, select **Use Default Labels**.

3. Click **OK** to return to the **ION Module Setup** dialog.

Customizing a setup register label

1. In the **ION Module Setup** dialog, select the setup register and click **Modify**.

2. Do one of the following:
   - To create a custom label, select **Use Custom Label** and then type a name for the label in the field.
   - To use the default label, select **Use Default Label**.

3. Click **OK** to return to the **ION Module Setup** dialog.

Resetting ION modules in a node diagram

Designer allows you to undo certain changes you have made in a node diagram since the last time you saved and sent your changes to the node. It refreshes the selected module icons so they reflect what is actually on the node. Reversible changes include:

- Changing the value of a setup register.
- Changing module or register labels.
- Changing the links between modules.

NOTE: Since adding and deleting modules happens immediately on a node, you cannot undo the deletion or the creation of a module.

To reset ION Modules:

1. Select the module(s) you want to reset.

2. Select **Edit > Reset**.

All the changes made to the selected modules since the last time you sent and saved the node diagram are discarded. Modules not included in the selection are unaffected (that is, if they had pending changes, the changes are still pending.)

Fast-linking ION modules in the Log Inserter

When linking modules in the Log Inserter, you can save time by using the Fast Linker utility. The Fast Linker automates much of the linking process, enabling you to quickly link any combination of event, data or waveform recorders. To access the Fast Linker options, press the CTRL key and click on the Log Inserter inputs.

For more information on using the Fast Linker utility, refer to the description for the Log Acquisition module in the *ION Reference*.

Configuring Security module

You can configure the device security through Designer. You can enable or disable security settings of the device. The device supports standard security and advanced security.

To enable standard security,

1. Open a node diagram of the device.

2. Select **Options**> **Change Standard Meter Security**.

   The **Change Standard Meter Security** window displays.

   > **NOTE**: If the **Change Standard Meter Security** option is disabled, refer to Changes in security settings of device for more information.

3. Clear the **Disable Standard Meter Password** check box and type the password in the **Password** field then re-type it in the **Confirm Password** field.

4. Click **OK**.

   A confirmation message displays.

5. Click **OK**.

6. Send and save the changes in Designer.

To enable advanced security,

1. Open a node diagram of the device.

2. Right-click the **Security Options** module icon to open the module setup.

3. Highlight the **SOP1 Enable Advanced Security** register and click **Modify**.

   > **NOTE**: If the **SOP1 Enable Advanced Security** register is not listed and the **SOP1 Legacy Security Support** register is listed, refer to Changes in security settings of device for more information.

4. Select **Enabled** in the options dropdown and click **OK** in the **Modify Enumerator Register** dialog.

5. Click **OK** in the **ION Module Setup** dialog to close it.

6. Add a new **Security Module** from the **ION modules** toolbox.

7. Right-click on the newly added **Security Module** icon to open the module setup.

8. Click **Change Password** button to open the **Change Advanced Meter Security** window.

9. Type the password in the **Password** field then re-type it in the **Confirm Password** field.

10. Click **OK**.

A confirmation message displays.

11. Click **OK** to acknowledge the message.

12. Click **OK** in the **ION Module Setup** dialog to close it.

13. Send and save the changes in Designer.

Copying and pasting ION modules

You can easily create a new function in a node diagram by copying and pasting modules. You can copy and paste a single module, a group of modules, grouping icons, text objects, or the entire node.

You can copy modules from one node and paste them into another, as long as the node you are pasting to supports the selected modules and has sufficient resources. When pasting modules, you can replace existing modules or add new ones.

You can also copy modules and save them as a framework. You and other users can then reuse this framework in other nodes. Saving a node's configuration as a framework provides a fast and easy way to program a large number of nodes at once.

NOTE: Persistent modules cannot be deleted, but they can be overwritten during a framework paste. Overwriting a persistent module effectively relinks its outputs and rewrites its label, but its core functionality remains intact (for example, the Master Reset module may be relabeled, but pulsing this module still initiates a master reset). When pasting a default framework onto a meter, use lock-paste to ensure that all persistent modules in the "old" default framework are overwritten by the corresponding persistent module in the "new" default framework. Persistent modules are listed in the "ION Device Template Reference".

Copying modules

Designer offers two ways to create copies of modules: copy and paste using the product's clipboard or copy or paste using a framework.

NOTE: The WinPM.Net Clipboard is a temporary storage area that holds any information cut or copied from Designer. This clipboard should not be confused with the Windows Clipboard.

When you copy a selection of ION modules, you can use **Copy**, **Cut**, or **Copy to Framework**:

- **Copy** saves the selection to the clipboard and leaves the original module(s) intact.
- **Cut** saves the selection to the clipboard and removes the original module(s).
- **Copy to framework** saves the selection as a file and leaves the original modules intact.

Additional considerations include pasting with shortcuts, modules that cannot be pasted, and cloning a node (copying and pasting an entire node configuration from one node diagram to another).

Cutting or copying ION modules to the clipboard

Use the Cut or Copy command to save a temporary copy of the selected modules to the clipboard. You can then use the Paste command to transfer the selection from the clipboard into any node diagram. Each copy retains the setup register values, custom labels, link information and layout of

the original selection.

> **NOTE**: When selecting modules to be cut or copied, you can also include grouping icons and text objects. If you select a grouping icon, all modules within the group are copied, as well as any nested grouping icons.

1. Select the module(s) you want to cut or copy.

2. Do one of the following:

   – To cut the selection to the clipboard and remove the original, select **Edit > Cut**.

   – To copy the selection to the clipboard and retain the original, select **Edit > Copy**.

   Designer copies the requested information to the clipboard.

After the selection has been copied to the clipboard, you can select **Edit > Paste** to paste it into any user diagram.

Copying ION modules to a framework

You can use the **Copy to framework** command to create a framework from the selected modules. A framework is a template that you can use to reproduce a group of modules. Unlike regular copying and pasting, a framework is stored as a file so you can paste it later.

When you select **Edit > Copy to framework**, Designer copies the layout, linking and configuration information of each module in the selected group. It saves the framework as an .fwn file. You can then use the **Paste from framework** command to paste the group in another node diagram.

You can use frameworks to simplify the process of programming multiple devices with similar configurations. For example, if you wanted to program a single function into several nodes, you could program the first node and then copy the configuration to a framework. You could then quickly program the other nodes by simply opening their node diagrams and pasting the framework you created. You can also use frameworks to program an entire node at once, considerably reducing the amount of time required to set up a large installation.

Before you save a group of modules as a framework, consider carefully which modules you want to include. If you include core modules in your framework, you will not be able to paste them unless you overwrite the existing core modules that correspond to those in your framework. Another approach is to select the modules "downstream" of the core module and save them as a framework. Then when you paste the framework into another node diagram, you can use the Maintain External Inputs option to preserve your links to the original core modules. (See Using the Paste Summary dialog for more information.)

> **NOTE**: After you have saved a framework, it can be used in another node diagram, as long as the node supports the modules included in the framework. Although it is possible to create a framework from one node type and use it with another node type, it is advisable to restrict your use of frameworks to nodes of the same type.

Copying a group of ION Modules to a framework

1. Select the group of module icons you want to copy.

2. Select **Edit > Copy to Framework** to open the **Copy to Framework** dialog.

3. In the File Name field, type a name for the framework, then click **Save**. The framework filename extension (.fwn) is added automatically and the selection is saved as a framework file.

After you have copied a group to a framework, you can select **Edit > Paste from framework** to paste it into a node diagram.

Pasting ION modules

You can paste a saved copy of one or more modules into any node diagram. Designer offers two ways to paste modules:

- **Paste** inserts a selection that has been placed on the clipboard using the **Cut** or **Copy** command

- **Paste from framework** inserts a selection that has been saved as a file.

Pasting modules is a little more complicated than copying them, because the action of adding modules to the node may require you to replace existing modules. Whether you choose **Paste** or **Paste from framework**, Designer displays a Paste Summary dialog that explains each aspect of the proposed changes.

You can use the Paste Summary box to define how to integrate the source modules into the destination modules. For example, if a source module is being pasted to a destination that is already occupied, you can either lock paste the new module over the existing one or free paste the new module into the next available destination.

If you paste a selection that contains a shortcut, Designer may paste the shortcut as a module. (See Pasting shortcuts for more information.)

In some cases, Designer may not paste every module in your selection. (See Modules that cannot be pasted for more information.)

Pasting ION modules from the clipboard or a framework

When pasting modules, the first thing to do is select the node diagram and window where you want the modules to go. If you have more than one window open, you can click on the title bar of the window you want to make it the active window.

> **NOTE**: You can paste modules into the same node diagram they were copied from or into a different one.

You can paste modules from two sources: the clipboard or a framework. When you paste a selection from either source, Designer describes the proposed changes in the Paste Summary dialog. You can use the Paste Summary area to review your changes and make modifications as necessary.

NOTICE

UNINTENDED DEVICE OPERATION

Do not use the lock paste option to overwrite existing destination modules.

Failure to follow these instructions can result in operational differences in required modules.

Only use the lock paste option if you are an advanced user familiar with ION architecture.

NOTE: The lock paste option overwrites an existing module but the outputs from that module are retained as inputs of other modules. Consequently, the module may not operate as expected. (See Using the Paste Summary dialog for more information.)

Pasting ION modules into a node diagram

1. Open the window where you want the modules to go. Click in the background of the window to clear any selected modules.

2. Do one of the following:

   – To paste a selection from the clipboard, select **Edit > Paste**.

   – To paste a selection from a framework, select **Edit > Paste from Framework**. In the Paste from Framework box, select the file you want and then click **Open**.

3. The **Paste Summary** dialog opens. Use the Paste Summary area to review the proposed changes (see Using the Paste Summary dialog, below, for more information). When you are satisfied with the summary, click **OK** to complete the procedure.

The selection is pasted into the active window. The entire selection is automatically selected so you can easily move it to a clear area in the window. All module icons in the selection appear as pending until you click **Send & Save** to save the changes to the node.

Using the Paste Summary dialog

Before Designer pastes a selection, it displays the **Paste Summary** dialog. Use the **Paste Summary** dialog to review and modify the proposed changes to the node diagram.

The **Paste Summary** dialog lists each module to be pasted and indicates its destination in the node. For each module, a short summary of the proposed modification as well as a more detailed description of how the module is pasted or why it cannot be pasted as requested displays in the dialog.
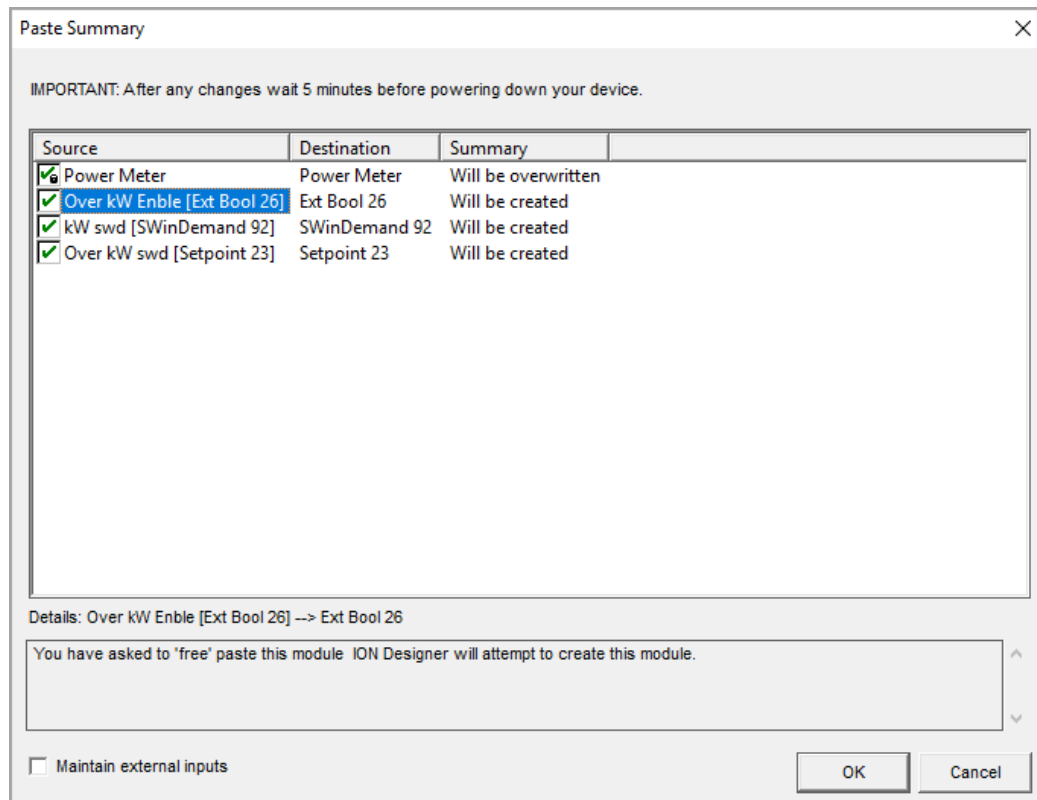
The following sections describe the various areas of the Paste Summary dialog.

Source

The Source column lists each module that was copied to the clipboard or framework. Each source module is listed by its default label unless a custom label has been specified (in which case the default name appears in brackets after the label). Select a module in this column to view additional information in the **Details** area at the bottom of the dialog.

Destination

The Destination column indicates where the copy of each source module will be placed in the node diagram. The module label listed in the Destination column may or may not match the module label listed in the Source column. (If the destination module is to be overwritten, it is reused; otherwise, the next available module is used.)

Summary

The Summary column indicates whether the source module is copied into the destination as a new module or if it replaces (overwrite) an existing module. This column also indicates if a module will not be pasted.

NOTE: You can sort the paste summary information alphabetically by source or destination, or by importance of summary. Click on the **Source**, **Destination**, or **Summary** column headers to sort the list by the chosen criteria.

Details

The Details area located at the bottom of the dialog provides more detail than the description in the Summary column. When you click on a module in the Source column, you can view a description of the proposed transaction in the Details box. This description explains how a module

is pasted or offers reasons why it will not be pasted.

> **TIP**: You can create a copy of the information displayed in the Details section of the Paste Summary box. To copy it, use the mouse to highlight the text you want, right-click the selection, then select **Copy**. You can then paste this selection into a word-processing program.

Maintain external inputs

The **Maintain External Inputs** check box allows you to paste the module selection with all external links intact. If this check box is cleared, all external links (links to modules outside the selection) are discarded.

The following example depicts an external link to a Power Meter module.
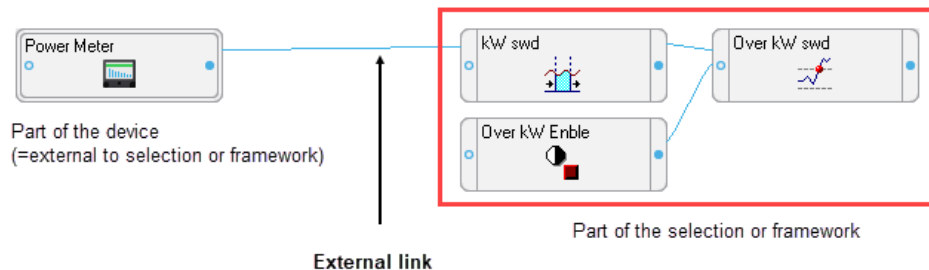
If you copied the modules on the right but omitted the Power Meter module from the selection, the link to the Power Meter module would be external. Upon pasting the selection, the external link to this module would be lost unless you selected "Maintain external inputs" on the Paste Summary box.

Check Boxes (Free Paste/Lock Paste/No Paste)

The check boxes to the left of each module indicate the proposed action (that is, how the module will be pasted):

- A green check mark ☑ indicates a free paste.

- A check mark with a lock symbol 🔒 indicates a lock paste.

- A red "X" symbol ☒ indicates the module will not be pasted.

- An "X" symbol without a check box ✖ identifies a module that has no possible destination and cannot be pasted.

You can click on a check box to change the proposed action. For example, if you click on a "free paste" ☑ or a "will not be pasted" ☒ symbol, it changes to a "lock paste" 🔒 symbol. When you change the type of paste, the descriptions in the Summary column and the Details box also change to describe the new action. (See Free paste or lock paste?, below, for more information.)

> **TIP**: To change multiple module check boxes in the paste summary at the same time, click the first check box in your list, hold down the **Shift** key, and then click the last check box in the list you want to change.

Free paste or lock paste?

You can use the free paste and lock paste options to instruct Designer how to respond to potential conflicts. For example, if you want to paste the module "Setpoint 1" into a diagram that already has a "Setpoint 1" module, you have two options. You can either add the new module under a new name (for example, "Setpoint 2") or you can overwrite the existing module with the new one and reuse the old name.

The first option in the above example is a free paste. It can be used to paste new modules into a diagram without deleting any existing modules. This is useful for integrating new functions into a node without disrupting the node's existing functionality. Although this may seem desirable, another option is available. By leaving existing modules intact, a free paste can result in needless duplication of functions. In addition, some groups of modules are interdependent and may require a lock paste to operate correctly.

The second option in the above example is a lock paste. It instructs Designer to recreate the source modules exactly, even if this requires Designer to overwrite existing modules in the diagram. When used to quickly replace old modules with new ones, lock pasting is an efficient alternative to the time-consuming process of deleting a module, adding a new one and then configuring it.

You can use a combination of both free paste and lock paste within a single paste operation.

NOTE: You can pre-select modules for Designer to reuse when free pasting. In the actual diagram where you paste the selection, press the SHIFT key and then click on any module icons that you don't want. When you paste the new selection, Designer has the option of replacing any of the selected modules with the new ones.

NOTE: Overwriting a module is not the same as deleting the module and then pasting a replacement module. If a module is overwritten, links from the outputs of that module are retained as the inputs of other modules. If a module is deleted, these links are also deleted and do not exist after the replacement module is pasted.

Pasting shortcuts

Identified by the symbol , a shortcut icon is a representation of a module icon that is located in another window.

You can copy a shortcut in the same way you copy the original module, but when you paste a shortcut, one of two things happens: It is either pasted as a shortcut to an existing module (if one is available) or as a new module. In either case, it continues to perform its function as before.

There is a basic reason for replacing a shortcut with a module: a shortcut cannot exist in a node without the module it represents. When you paste a selection containing a shortcut, Designer searches both the selection to be pasted and the destination to locate the module the shortcut represents. If it finds the module, it pastes the shortcut as a shortcut. If it doesn't find the module, it replaces the shortcut with a copy of the module. In this way, Designer functions so that the pasted selection is complete and not dependent on a module that is unavailable in the destination node.

Designer performs this process automatically; it requires no input from you.

Modules that cannot be pasted

Some modules cannot be pasted because their settings are non-transferable between nodes. All Factory, Communications, and Diagnostics modules must be individually configured for each node.

When pasting a selection that contains a Factory, Communications or Diagnostics module, Designer pastes the location of the icons only.

Cloning a node

You can quickly configure multiple nodes by cloning a node. Essentially, cloning a node involves copying a node's entire configuration to a framework then pasting it onto another node. This effectively reprograms the node receiving the framework as an exact copy of the original.

> **NOTE**: When you paste an entire node's configuration, the Factory, Communications, and Diagnostics modules are not pasted because they could disrupt the operation of the device's communications. If you need to change the settings of these modules, you can do so manually.

Copying a node's entire configuration

1. Open the node diagram of the node you want to copy. Ensure that you are in the main or root window of the diagram.

2. Select **Edit > Select All**. All core modules, standard modules, grouping windows and text icons are selected.

3. Select **Edit > Copy to framework** to open the **Copy to framework** dialog.

4. In the File name box, type a name for the framework and then click **Save**. The framework filename extension (.fwn) is added automatically and the entire node configuration is saved in this framework file.

After you have saved the source node's configuration as a framework, you can paste it to other nodes of the same type. This file can also serve as a backup in case you want to restore the configuration of the source node.

Pasting a node's entire configuration into another node

1. Open the node diagram of the node to which you want to paste the configuration. Ensure that you are in the main or root window of the diagram.

2. Select **Edit > Select All**. Press the **Delete** key to remove the node's existing configuration. A dialog opens listing the standard modules that will be deleted and the core modules that cannot be deleted. Click **OK** to continue.

   The standard modules and text icons are deleted, and the core modules (and any grouping windows containing core modules) remain.

3. Select **Edit > Select All** to select the remaining module (and grouping) icons.

4. Select **Edit > Paste from framework** to open the **Paste from framework** dialog, select the framework file that contains the node configuration you want to paste, and then click **Open**.

   The **Paste Summary** dialog opens. (See Using the Paste Summary dialog for information on using the Paste Summary box.)

5. Select all modules in the list (click last item, hold the SHIFT key, and then click first list item - the entire list appears highlighted).

6. Change the paste type to lock paste (hold the SHIFT key and then click the paste type check box until all modules are set to lock paste ☑🔒).

7. Click **OK** to complete the procedure.

The selected framework is pasted into the node diagram. All module icons in the selection appear as pending until you select **Send & Save** to save the changes to the node.

Viewing output registers, setup registers, and inputs

Each module icon has an input symbol and an output symbol. The input symbol (on the left) provides access to the module's inputs and the output symbol (on the right) provides access to the module's output and setup registers.

A	Input
B	Output

A register is a type of memory bank that stores a module's linking and configuration information. Each module uses two types of registers:

*   Output registers store data that has been processed by the module (such as numeric values, event log entries, and waveform data).

*   Setup registers store the module's configuration information.

Both types of registers serve as storage locations for the module.

Viewing output register information

Output register information includes the module's available output registers, the real-time values of the output registers and the owners of the output registers. Each of these can be accessed from the output symbol located on the right side of the module icon.

A	Click the output symbol to access the module's output register information

Viewing available output registers

Click the output symbol.

Viewing the real-time values of output registers

Press the SHIFT key and click the output symbol.

Viewing the owners of output registers

Right-click the output symbol. This displays an output register's owners so you can identify all dependent modules on the node and determine the purpose of each dependent module. This is useful when determining the effects and consequences of deleting a module.

Viewing setup register information

A module's setup registers contain its configuration information. When a module processes data, it manipulates the data according to the guidelines stored in its setup registers. Setup register information includes the module's available setup registers, the setup register settings, and the setup register owners. Each of these can be accessed from the output symbol located on the right side of the module icon.
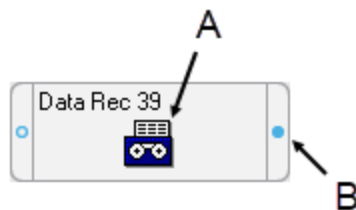
A	Right-click the graphic to access the ION Module Setup dialog.
B	Press CTRL and click the output symbol to access the ION Module Setup dialog. Press CTRL+SHIFT and click the output symbol to display the ION Module Setup dialog to view Setup Register settings. Press CTRL and right-click the output symbol to view SetupRegister owners.

Viewing setup registers

Press the CTRL key and click the output symbol or right-click the graphic in the center of the module to display the ION Module Setup dialog.

Viewing setup register settings

Press CTRL+SHIFT and click the output symbol or right-click the graphic in the center of the module to display the ION Module Setup dialog.

Viewing setup register owners

Press the CTRL key and right-click the output symbol.

Viewing input information

Input information includes the module's inputs and a sample of its real-time input values. Each of these can be accessed from the module's input symbol.

A	Click the input symbol to access the module's input information

Viewing inputs

Click the input symbol.

Viewing real-time input values

Press the SHIFT key and click the input symbol.

Chapter 6: Virtual Processor service (VIP)

The Virtual Processor is a service that operates on the WinPM.Net server, providing coordinated data collection, data processing, and control functions for groups of meters. This makes distributed operations possible, providing customized solutions to a variety of industrial, commercial, and power utility needs.

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use control objects for time-critical functions because delays can occur between the time a control action is initiated and when that action is applied.
- Do not create a custom control object in Virtual Processor for remote equipment operation without configuring it with the appropriate access level, and without configuring a status object to indicate the status of the control operation.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

The Virtual Processor lets you gather the information available from your network of power monitoring devices, and enables you to categorize, manipulate, and/or customize the data before distributing the information to the different departments in your company. You could think of the Virtual Processor as a virtual device, capable of collecting and processing data from several power monitoring devices, analyzing the information and performing control functions as required. The Virtual Processor's name implies its characteristics:

- **Virtual** - The Virtual Processor runs in the memory of your PC, not as a remote device.

- **Processor** - The Virtual Processor contains a wide selection of ION modules, which it uses to process information.

Using the Virtual Processor

A **VIP.DEFAULT** Virtual Processor service is created by default. However, this Virtual Processor is not configured at the factory. To perform data collection, data analysis, Modbus communications, or distributed control, you must first create ION module frameworks, using Designer. Refer to the *ION Reference* for information on ION modules.

Configuring the Virtual Processor with Designer

Use Designer to perform all Virtual Processor configuration, including adding new modules, configuring module setup registers, and linking module inputs and output registers to other modules.

Creating a second Virtual Processor

Create a second WinPM.Net Virtual Processor Service and start it using the following steps:

1. Open a command prompt window.

2. Register the Virtual Processor as a service from the ...\WinPM.Net\system\bin folder by typing: **vip.exe -Service -N<custom Virtual Processor name>**

3. Open **Control Panel > Administrative Tools > Services** and start the new registered Virtual Processor.

Viewing Virtual Processor data with Vista

After you have used Designer to add and link modules in the Virtual Processor, you can create a user diagram in Vista and link it to the modules in the Virtual Processor. You can then use your user diagram to display system data, monitor alarms, and control basic functions in your Virtual Processor framework.

Configuring the Virtual Processor to interact with the desktop

1. Open **Control Panel > Administrative Tools > Services**.

2. Right-click the WinPM.Net Virtual Processor Service and select **Properties**.

3. Select the **Log On** tab.

4. Select the "Allow service to interact with desktop" check box and click **OK**.

5. Stop and restart the WinPM.Net Virtual Processor Service.

Distributed control

Virtual Processor Distributed Control modules are a means of transferring information between the different devices in your network. These modules can be used to build frameworks for control processes such as alarm annunciation, plant-wide demand control, power factor control, load shedding, paging, and generator switching.

Before you decide to use a Virtual Processor to implement a control system, it is important to understand the various factors that influence the speed with which the Virtual Processor is able to generate a control action. Some of these factors include network traffic, the number of polled devices in your network, the reliability of your communications network, and the Virtual Processor workstation's CPU usage.

⚠ WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use control objects for time-critical functions because delays can occur between the time a control action is initiated and when that action is applied.

- Do not create a custom control object in Virtual Processor for remote equipment operation without configuring it with the appropriate access level, and without configuring a status object to indicate the status of the control operation.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

The information presented in this section assumes that you are familiar with the various ION modules and their functions. Refer to the *ION Reference* for details on Distributed Control and other modules.

Distributed control modules

The Virtual Processor uses four modules to implement Distributed Control frameworks. Three types of Distributed Control modules (Distributed Boolean, Numeric, and Pulse modules) receive data from devices or other Virtual Processors within your network and transfer this data to the node address specified in the module's setup registers. The Data Monitoring module is used as a means of disabling a control module, if the data at the source input (which is used to make the control decision) becomes "stale".

Distributed control module setup registers

All Distributed Control modules (Distributed Boolean, Distributed Numeric, and Distributed Pulse modules) have a Source input and an Activate input. The Activate input must be linked to another module, and it must be on for a Distributed Control module to function. These modules also contain a *Destination* setup register that specifies the node receiving the data.

Distributed Boolean module

All Distributed Boolean modules have one input that must linked to an External Boolean output register. Whenever this register's Boolean value changes state, the new value is sent to the node defined in the *Destination* setup register.

Distributed numeric module

A Distributed Numeric module's input must be linked to an External Numeric module. This input's value is sent to the node specified in the Destination setup register. This module has two additional setup registers that are used to determine when and how often new data is sent to the *Destination* node.

Distributed pulse module

All Distributed Pulse modules have one input that must link to an External Pulse module. This module's pulse is sent to the node defined in the *Destination* setup register.

Data monitoring module

The Data Monitor module provides a means of alerting you to communication problems that may occur between the Virtual Processor and any ION node referenced by the Data Monitor's *Source* inputs.

> **NOTE**: ACCESS meters with Advanced Security enabled do not accept control operations from the Virtual Processor. In order to enable these control operations, you need to provide a device password in Management Console for these applications to use. See Accessing meters with security for information on configuring Advanced Security.

Distributed control network performance

It is important to maximize the speed at which the Virtual Processor receives the data used to make control decisions. The performance of a Virtual Processor using Distributed Control is sensitive to factors such as network traffic, the Virtual Processor's global parameters, and the workstation's CPU usage.

The following sections provide some guidelines for improving a distributed control network's performance.

System configuration

- Minimize the number of devices per site for sites including devices used for distributed control.

- Minimize the number of applications (such as Vista diagrams) that are requesting data from the control site.

- Decrease the Virtual Processor's Client Polling Period.

- Maximize the site's baud rate.

CPU performance

- Dedicate a single Virtual Processor for control purposes; an additional Virtual Processor can be run for less critical functions.

- Do not run CPU intensive functions on the same workstation as your distributed control Virtual Processor.

Distributed control applications

The Virtual Processor's Distributed Control frameworks can perform various control processes; two examples are presented below. Ensure that you are familiar with the sections above before using any of these module frameworks in a control system.

Example: Passing a value between devices

The following example shows how the numeric output of the Integrator modules in meter "A" is sent to an External Numeric module in meter "B".
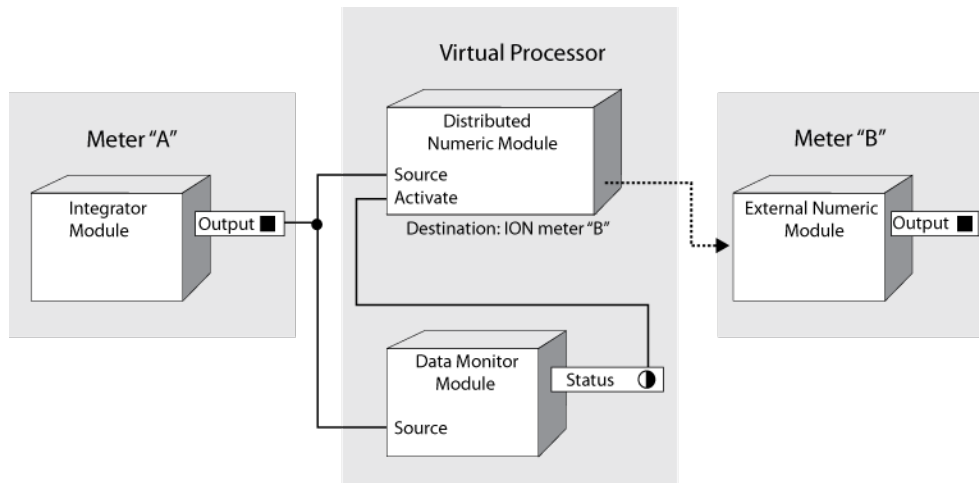
Note that the Distributed Numeric module is disabled if the Data Monitor module detects a communication problem between the meter "A" and the Virtual Processor (for example, timeouts or slow network connections). As a result, control actions are only performed based on up-to-date data.

Example: Device control

This example framework monitors total harmonic distortion (THD), which could cause a piece of equipment (like a power transformer) to overheat. When a setpoint is reached, the system sends a network message to a workstation and turns on a fan.
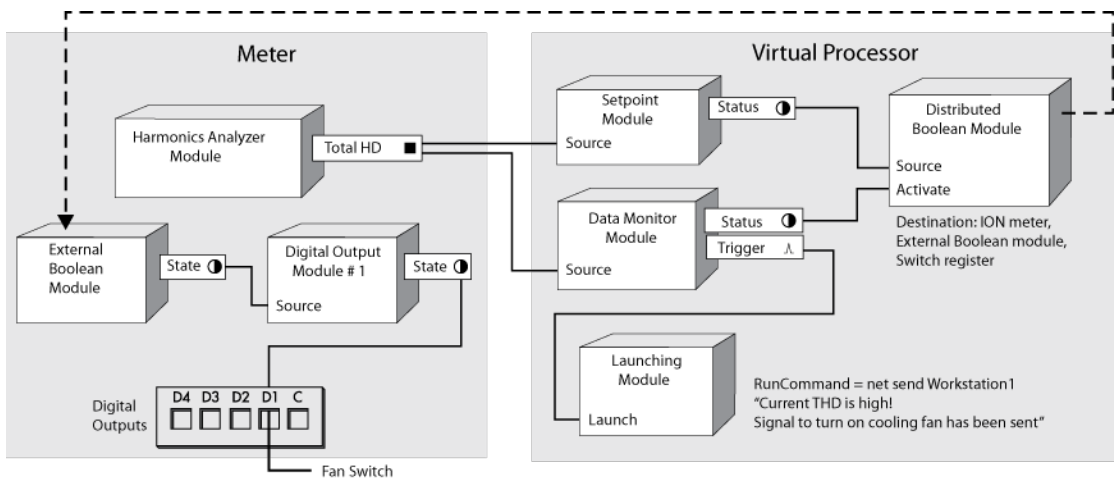
Example: Data aggregation

In the following example, meters monitor average current values (I avg). Every day at a specified time (determined by the Periodic Timer module), the Data Recorder Module stores the maximum average current value for each meter.
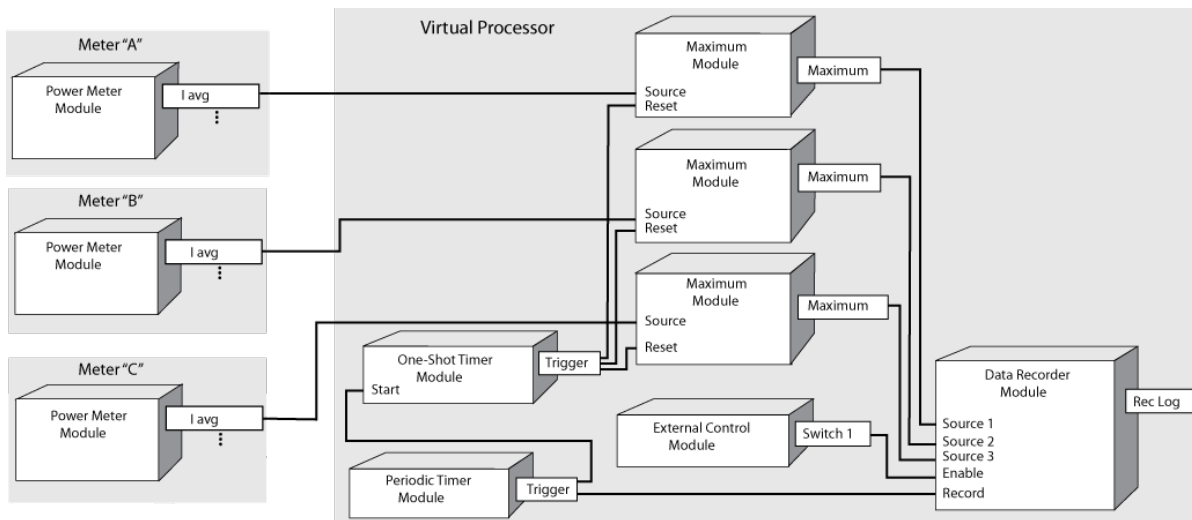
Example: Logging data from meters

The Virtual Processor can be used to log data from meters that do not have Data Recorders. In the following example, the total kWh from a meter's Integrator module is recorded every 15 minutes by a Virtual Processor's Data Recorder module.

Note that the Virtual Processor holds only 100 records at a time, that is, it holds about one day of 15 minute data. If the Virtual Processor stops operating, data logging also stops.
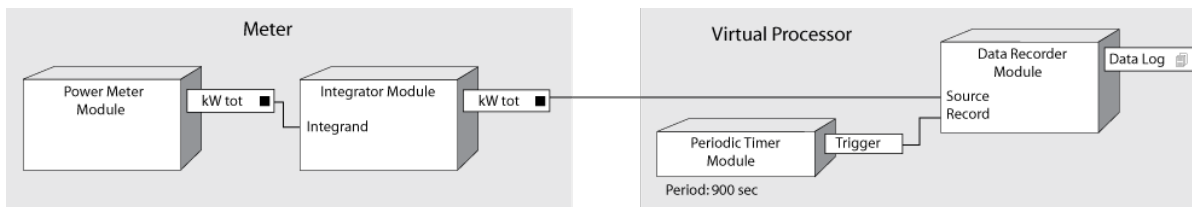
Setting global parameters

The Virtual Processor global operating parameters and Modbus network setup are configured using Virtual Processor Setup.

The global parameters do not need to be changed for normal operation. The Virtual Processor operates properly in most applications with the default settings. See Modifying the Global Parameters for details.

To set up the Virtual Processor's Modbus network, see Virtual Processor setup.

NOTE: The WinPM.Net Virtual Processor Service must be restarted for any changes to take effect, as the Virtual Processor only reads the vipsys.dat file once on startup. The WinPM.Net Virtual Processor Service can be restarted, like any other service, using the Services window found via the Control Panel service applet.

Chapter 7: Hierarchy Configuration Utility

The Hierarchy Configuration Utility is intended for Application Engineers or advanced users working with hierarchies in WinPM.Net.

The utility is a command-line executable that allows you to save, restore, or add hierarchy content.

The primary purpose of the utility is to give you a way to add a large number of entries to a hierarchy rather than creating individual entries in Hierarchy Manager. This task consists of:

- Saving a skeleton hierarchy configuration in a CSV file.

- Editing the CSV file to associate the devices listed in the file to the nodes of the hierarchy.

- Populating Hierarchy Manager with the updated contents of the hierarchy configuration CSV file.

The utility also provides options for:

- Saving or replacing a hierarchy configuration currently defined in Hierarchy Manager.

- Installing or saving a hierarchy template.

- Refreshing a configuration cache to ensure that all devices are included when you save a hierarchy configuration.

- Deleting a hierarchy configuration prior to restoring a saved configuration.

Throughout this document, references to "configuration" and "template" should be interpreted as follows:

- **configuration** – the entries representing the node types, devices, and virtual meters that comprise the content of a hierarchy.

- **template** – defines the node names, attributes, relationships, and views in Hierarchy Manager.

Hierarchy Configuration Utility command and options

Run the Hierarchy Configuration Utility command, `HierarchyConfig.exe`, from a Command Prompt.

To open a Command Prompt window:

1. Use Windows Explorer to navigate to:

   `...\WinPM.Net\Applications.`

2. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

The command syntax is:

`HierarchyConfig.exe` *option* `<file path>\<file name>`, where:

- *option*: see Command options below.

- `<file path>`: the location of the file that the command runs on.

- `<file name>`: the name of the CSV or XML file, including the `.csv` or `.xml` file extension, respectively.

Command options

Use the command as follows (type the command and press **Enter**):

Running the command without options

- To see a brief description of available command options:

```
HierarchyConfig.exe
```

Running the command using the `config` option

- To create a skeleton configuration CSV file that you can use to define a hierarchy configuration:

```
HierarchyConfig.exe export config <file path>\<file name>.csv
```

Note that this process creates a header row in the CSV file with percentage, node type names, and virtual meter column headings. It also includes a list of the devices in the system that are **not** included in a hierarchy. This allows you to specify which devices you want to include in the hierarchy when you import the configuration.

Optional: Including all devices

Add the `alldevices` option at the end of the export config command to create a CSV file that lists all of the devices, whether or not they are included in a hierarchy. This option ensures that apportioned devices or devices that constitute a virtual meter are included in the file.

To include all devices, type the following:

```
HierarchyConfig.exe export config <file path>\<file name>.csv
alldevices
```

Note that the CSV file created with the `alldevices` option is intended for reference purposes only and not for updating an existing hierarchy.

- To add the contents of the updated skeleton configuration CSV file to Hierarchy Manager:

```
HierarchyConfig.exe import config <file path>\<file name>.csv
```

NOTICE

LOSS OF DATA

- Do not use the hierarchy configuration CSV file to incrementally update an existing hierarchy.
- Use the Hierarchy Manager application to incrementally update an existing hierarchy.

Failure to follow these instructions can result in the corruption of hierarchy data.

Optional: Adding a timestamp to configuration entries

You can also add a timestamp to the configuration entries. Adding year, month, and day information adds a start date to the imported references corresponding to the date entered. If no date information is provided, the start date of the hierarchy information is the system start date.

To add a timestamp, type `HierarchyConfig.exe import config <file path>\<file name>.xml [yyyy/mm/dd]` where `[yyyy/mm/dd]` is the year, month and day that the imported hierarchy references use as a start date.

- To back up a complete representation of the hierarchy and all virtual meters in an XML file:

```
HierarchyConfig.exe export config <file path>\<file name>.xml
```

This creates a backup of the hierarchy before you modify the hierarchy entries in Hierarchy Manager.

- To restore a hierarchy configuration from a backup configuration XML file:

```
HierarchyConfig.exe import config <file path>\<file name>.xml
```

NOTICE

LOSS OF DATA

- Do not use the hierarchy configuration XML file to incrementally update an existing hierarchy.
- Use the Hierarchy Manager application to incrementally update an existing hierarchy.

Failure to follow these instructions can result in the corruption of hierarchy data.

This process deletes the current hierarchy configuration defined in Hierarchy Manager and replaces it with the hierarchy configuration in the XML file.

- To refresh a configuration cache to ensure that all devices are included in the saved configuration:

```
HierarchyConfig.exe refresh config
```

- To delete a hierarchy configuration:

```
HierarchyConfig.exe delete config
```

Only use this process if you have previously saved your configuration.

Running the command using the `template` option

- To save the hierarchy template used in Hierarchy Manager:

```
HierarchyConfig.exe export template <file path>\<file name>.xml
```

The template defines the node names, attributes, and relationships used in Hierarchy Manager.

- To replace an existing hierarchy template used in Hierarchy Manager with a different one:

```
HierarchyConfig.exe import template <file path>\<file name>.xml
```

Populating a hierarchy

After installing and commissioning WinPM.Net, you can populate Hierarchy Manager by associating devices to nodes in a skeleton configuration CSV file and importing the entries into Hierarchy Manager. Following a successful import, you can use Hierarchy Manager to modify your hierarchy information. You can then use hierarchies in other WinPM.Net applications, such as in view-enabled reports in the Web-based Reports application, and in the Dashboards application.

For illustration purposes, the default hierarchy template in Hierarchy Manager is used as a reference for this task.

The skeleton configuration CSV file created from the default hierarchy template used in Hierarchy Manager includes a header row with column headings of **Devices_Full Name**, **_Percentage**, **Site_Name**, **Buildings_Name**, **Areas_Name**, and **Virtual Meter_Name**. It also includes a list of the devices in the system that are **not** included in a hierarchy so that you can associate the devices to the hierarchy nodes.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1						
3	Test_Site.Device2						
4	Test_Site.Device3						
5	Test_Site.Device4						
6	Test_Site.Device5						
7	Test_Site.Device6						
8	Test_Site.Device7						
9	Test_Site.Device8						
10	Test_Site.Device9						
11							

Different hierarchy templates can be used in Hierarchy Manager. Each hierarchy template determines the information that is included in the configuration CSV file. For example, the `Site Building Area Tenant.xml` hierarchy template file results in column headings of **Devices_Full Name**, **_Percentage**, **Site_Name**, **Buildings_Name**, **Areas_Name**, **Tenants_ Name**, **Tenants_Contact**, **Tenants_email**, **Tenants_Phone**, and **Virtual Meter_Name** in the configuration CSV file.
(Sample and generic hierarchy templates are located in the:
`...\Applications\HierarchyManager\SampleTemplates` folder.)

The procedure for updating a skeleton configuration CSV file follows a consistent pattern regardless of the template used in Hierarchy Manager.

The process for adding a large number of entries to a hierarchy consists of:

- Saving a skeleton hierarchy configuration in a CSV file.

- Editing the CSV file.

- Updating Hierarchy Manager with the new entries and verifying the outcome.

<table>
<tr><td align="center">

NOTICE

</td></tr>
</table>

LOSS OF DATA

- Do not use the hierarchy configuration CSV file to incrementally update an existing hierarchy.

- Use the Hierarchy Manager application to incrementally update an existing hierarchy.

Failure to follow these instructions can result in the corruption of hierarchy data.

Before you begin

Before you export the skeleton configuration CSV file from Hierarchy Manager, make sure that you complete the following tasks:

- Plan how the hierarchy is going to be configured, such as: naming conventions, which devices are going to be mapped to which Node, and how the different elements are grouped together. This can save time when you enter content in the configuration CSV file and it is very important when dealing with a large number of devices.

  For illustration purposes in this task, devices are going to be mapped to a physical layout consisting of 4 areas within a building.
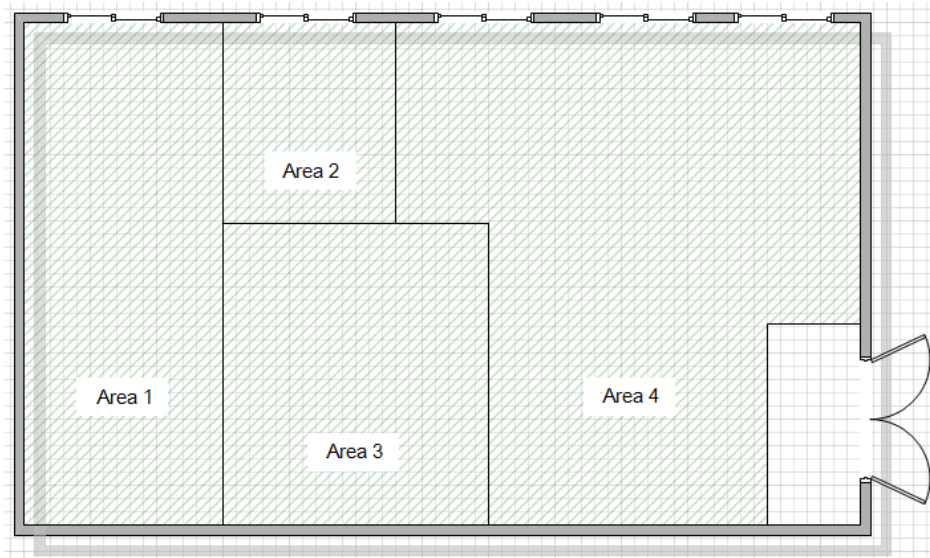
- Install Microsoft Excel, or another spreadsheet application that you can use to open and edit CSV files on your system.

- Add devices or logical devices to WinPM.Net. You can add additional devices at a later time, but you need to have an initial number of devices before you export the skeleton configuration CSV file.

Saving a hierarchy configuration in a CSV file

This process creates a skeleton hierarchy configuration CSV file with a header row that includes percentage, node type names, and virtual meter column headings. It also includes a list of the devices in the system that are **not** included in a hierarchy. This allows you to specify which of

those devices you want to include in the hierarchy when you import the contents of the hierarchy configuration CSV file.

If devices have recently been added, use the refresh command to update the configuration cache before proceeding. (See Refreshing the hierarchy cache for more information.)

1. Open a Command Prompt window:

    a. Use Windows Explorer to navigate to:

        `...\WinPM.Net\Applications.`

    b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe export config <file path>\<file name>` and press **Enter**.

    Where `<file path>` is the location for the exported CSV file, and `<file name>` is the name of the file, including the `.csv` file extension. If you do not specify a file path, the current directory is used.

    > **NOTE**: If you include spaces in the CSV file name and/or file path, you must enclose the file path and file name in quotation marks. For example, `HierarchyConfig.exe export config "C:\test file.csv".`

    The items in the skeleton hierarchy configuration CSV file includes a list of devices **not** included in a hierarchy, the heading **_Percentage**, headings for the node types, and the heading **Virtual Meter_Name**.

Adding content to the configuration CSV file

The following steps describe:

- Defining the main elements (instances of a node) of the hierarchy.

- Associating devices to areas.

- Adding a virtual meters and associating devices.

Defining the main elements of the hierarchy

Define the site, building, and area names in the file as follows:

1. Open the hierarchy configuration CSV file in Excel.

2. On an empty row below the list of devices, type the names that you want to use for the site, the building, and an area association in their respective columns.

As indicated in the previous description of the physical layout, there are 4 building areas. You now need to define the association of the 3 remaining areas to the building and site.

3. Repeat the previous step for each of **Area 2**, **Area 3** and **Area 4**, as shown in the following image.

This completes the association of the 4 areas to the building instance, **Building 1**, and the site instance, **Test Site**.

If there are additional buildings on the site, you can repeat the pattern that you entered. For example, if there is a Building 2 on the site, your entries would include the same site name, **Test Site**, but the building name is **Building 2**. Note that you should use unique names for the areas. For example, for **Building 2**, define the area names like **B2-Area 1**, **B2-Area 2**, and so on.

Associating devices to areas

Complete the following steps to associate devices to specific areas:

1. Type the area name that you defined in the previous set of steps, in the **Areas_Name** column on the row identifying the device full name.

   For example, to associate device **Test_Site.Device1** to **Area 1**, type **Area 1** in the **Areas_Name** column in that row.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2						
4	Test_Site.Device3						
5	Test_Site.Device4						
6	Test_Site.Device5						
7	Test_Site.Device6						
8	Test_Site.Device7						
9	Test_Site.Device8						
10	Test_Site.Device9						
11							
12			Test Site	Building 1	Area 1		
13			Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

2. Continue to associate any of the remaining devices to their respective areas. Note that you can assign multiple devices to the same area. For example, the following image shows two devices assigned to **Area 3** and two to **Area 4**.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2				Area 2		
4	Test_Site.Device3						
5	Test_Site.Device4						
6	Test_Site.Device5				Area 2		
7	Test_Site.Device6				Area 3		
8	Test_Site.Device7				Area 3		
9	Test_Site.Device8				Area 4		
10	Test_Site.Device9				Area 4		
11							
12			Test Site	Building 1	Area 1		
13			Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

Adding a virtual meter and associating devices

To define a virtual meter:

1. Type the virtual meter name in the **Virtual Meter_Name** column on the row identifying each device full name that you want to associate with the virtual meter.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2				Area 2		
4	Test_Site.Device3					VM1	
5	Test_Site.Device4					VM1	
6	Test_Site.Device5				Area 2		
7	Test_Site.Device6				Area 3		
8	Test_Site.Device7				Area 3		
9	Test_Site.Device8				Area 4		
10	Test_Site.Device9				Area 4		
11							
12			Test Site	Building 1	Area 1		
13			Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

2. To apportion a percentage of a device's value to the virtual meter, type a percent value for the applicable device in the **_Percentage** column. For example, apportion 25% of **Test_Site.Test_8600_C** and 75% of **Test_Site.Test_8600_D** to virtual meter **VM1**.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2				Area 2		
4	Test_Site.Device3	25				VM1	
5	Test_Site.Device4	75				VM1	
6	Test_Site.Device5				Area 2		
7	Test_Site.Device6				Area 3		
8	Test_Site.Device7				Area 3		
9	Test_Site.Device8				Area 4		
10	Test_Site.Device9				Area 4		
11							
12			Test Site	Building 1	Area 1		
13			Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

3. To associate a virtual meter to a specific area, add the virtual meter name to the **Devices_Full Name** column on a row defining the area. For example, to associate **VM1** to **Area 2**, add **VM1** to line **13** (in this example) in the CSV file.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2				Area 2		
4	Test_Site.Device3	25				VM1	
5	Test_Site.Device4	75				VM1	
6	Test_Site.Device5				Area 2		
7	Test_Site.Device6				Area 3		
8	Test_Site.Device7				Area 3		
9	Test_Site.Device8				Area 4		
10	Test_Site.Device9				Area 4		
11							
12			Test Site	Building 1	Area 1		
13	VM1		Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

You can also associate a virtual meter (**VM1**) to another virtual meter by including the virtual meter name in the **Devices_Full Name** column and adding the additional virtual meter name on the same row under the **Virtual Meter_Name** column.

	A	B	C	D	E	F	G
1	Devices_Full Name	_Percentage	Site_Name	Buildings_Name	Areas_Name	Virtual Meter_Name	
2	Test_Site.Device1				Area 1		
3	Test_Site.Device2				Area 2		
4	Test_Site.Device3	25				VM1	
5	Test_Site.Device4	75				VM1	
6	Test_Site.Device5				Area 2		
7	Test_Site.Device6				Area 3		
8	Test_Site.Device7				Area 3		
9	Test_Site.Device8				Area 4		
10	Test_Site.Device9				Area 4		
11	VM1					VM2	
12			Test Site	Building 1	Area 1		
13	VM1		Test Site	Building 1	Area 2		
14			Test Site	Building 1	Area 3		
15			Test Site	Building 1	Area 4		
16							

When you complete your updates in the hierarchy configuration CSV file, you can now update Hierarchy Manager by importing the contents of the CSV file.

Adding your entries to Hierarchy Manager

Complete the following steps to add the contents of the hierarchy configuration CSV file to Hierarchy Manager:

1. Open a Command Prompt window:

    a. Use Windows Explorer to navigate to:

        `...\WinPM.Net\Applications.`

    b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe import config <file path>\<file name>` and press **Enter**.

    Where `<file path>` is the location of the file you are importing, and `<file name>` is the name of the file, including the `.csv` file extension. If you do not specify a file path, the current directory is used.

    The import process provides feedback during each stage of the import. If the import operation is unsuccessful, the utility provides information about possible reasons for the unsuccessful operation.

    > **NOTE**: If the CSV file name and/or file path contains spaces, you must enclose the file path and file name in quotation marks. For example, `HierarchyConfig.exe import config "C:\test file.csv"`.

3. Verify the correct hierarchy configuration in Hierarchy Manager.

Saving and restoring a hierarchy configuration

The Hierarchy Configuration utility allows you to save (export) the configuration as an XML file. The file contains a full representation of the current hierarchy in Hierarchy Manager, including:

- Nodes.

- Relationships between nodes, including their time dimensions.

- All apportionment nodes.

This lets you back up the hierarchy before modifying it in the Hierarchy Manager application. You can import the hierarchy configuration saved in the XML file to restore your hierarchy in Hierarchy Manager.

> **NOTE**: Restoring (importing) a configuration XML file deletes the current hierarchy configuration and replaces it with the one represented in the file.

NOTICE

LOSS OF DATA

- Do not use the hierarchy configuration XML file to incrementally update an existing hierarchy.
- Use the Hierarchy Manager application to incrementally update an existing hierarchy.

Failure to follow these instructions can result in the corruption of hierarchy data.

Saving the entire hierarchy configuration in an XML file

To save a configuration XML file:

1. Open a Command Prompt window:

   a. Use Windows Explorer to navigate to:

      `...\WinPM.Net\Applications.`

   b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Run `HierarchyConfig.exe export config <file path>\<file name>`, where `<file path>` is the location where you want to save XML file, and `<file name>` is the name of the XML file, including `.xml` as the file extension. If you do not specify a file path, the current directory is used.

   NOTE: If the XML file name and/or file path contains spaces, you must enclose the file path and file name in quotation marks. For example, `HierarchyConfig.exe export config "C:\test file.xml".`

Restoring the entire hierarchy configuration from an XML file

NOTE: Restoring a configuration XML file deletes the current hierarchy configuration and replaces it with the one represented in the file.

NOTICE

LOSS OF DATA

- Do not use the hierarchy configuration XML file to incrementally update an existing hierarchy.
- Use the Hierarchy Manager application to incrementally update an existing hierarchy.

Failure to follow these instructions can result in the corruption of hierarchy data.

To restore the configuration XML file:

1. Open a Command Prompt window:

    a. Use Windows Explorer to navigate to:

        `...\WinPM.Net\Applications.`

    b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe import config <file path>\<file name>`, where `<file path>` is the location of the file you are importing, and `<file name>` is the name of the XML file, including `.xml` as the file extension, to be imported. If you do not specify a file path, the current directory is used.

3. Verify the correct hierarchy configuration in Hierarchy Manager.

Installing or saving a hierarchy template

The `import template` command option lets you install a hierarchy template as a replacement for the template used in Hierarchy Manager. The `export template` command option lets you save a copy of the hierarchy template used in Hierarchy Manager. Prior to using these commands, you should back up your existing hierarchy configuration (see Saving the entire hierarchy configuration in an XML file).

Installing a hierarchy template

Use the import template option to replace the template currently installed in your system with a different one.

NOTICE

LOSS OF DATA

The new template overwrites the existing template, which permanently removes all information contained in the original. Export the existing hierarchy template from WinPM.Net before importing a new template.

Failure to follow these instructions can result in a loss of data.

When you import the hierarchy template XML file, the Hierarchy Configuration Utility performs a validation test on the file. If the file validation is not successful, the utility does not import the file and provides information indicating where the validation process stopped. Check for any discrepancies in the file, save the changes and reimport the file.

To import a hierarchy template XML file:

1. Open a Command Prompt window:

    a. Use Windows Explorer to navigate to:

        `...\WinPM.Net\Applications.`

b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe import template <file path>\<file name>`, where `<file path>` is the location of the XML file, and `<file name>` is the name of the XML file, including `.xml` as the file extension, to be imported. If you do not specify a file path, the current directory of the Hierarchy Configuration Utility is used.

   The utility provides feedback for each stage of the import. If the import operation is unsuccessful, the utility provides information about why the import is unsuccessful.

   NOTE: If the XML file name and/or file path contains spaces, you must enclose the file path and file name in quotation marks. For example, `HierarchyConfig.exe import template "C:\test file.xml"`.

3. Verify that the template has been successfully imported in Hierarchy Manager.

   Hierarchy Manager should now display the nodes defined in the newly imported template.

Saving a hierarchy template

Use the export template option to save the Hierarchy Manager template currently installed in the system as an XML file. The file defines the node names, attributes, relationships, and views for Hierarchy Manager.

To export a hierarchy template XML file:

1. Open a Command Prompt window:

   a. Use Windows Explorer to navigate to:

      `...\WinPM.Net\Applications`.

   b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe export template <file path>\<file name>`, where `<file path>` is the location where you want to save the XML file, and `<file name>` is the name of the XML file, including `.xml` as the file extension. If you do not specify a file path, the current directory is used.

   NOTE: If the XML file name and/or file path contains spaces, you must enclose the file path and file name in quotation marks. For example, `HierarchyConfig.exe export template "C:\test file.xml"`.

3. Navigate to the location you specified to confirm that the XML file is successfully saved.

Refreshing the hierarchy cache

Use the refresh config option to refresh the configuration cache. If you recently added new devices to WinPM.Net , use this command before you export a hierarchy configuration CSV file so that an up-to-date list of devices is included in the file. If the refresh configuration is not run, there is a possibility that devices recently added to WinPM.Net may not be present in the device list in the hierarchy configuration CSV file. The best practice is to run the refresh config command before exporting any CSV or XML file from Hierarchy Manager.

To refresh the configuration cache:

1. Open a Command Prompt window:

   a. Use Windows Explorer to navigate to:

      `...\WinPM.Net\Applications.`

   b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Run `HierarchyConfig.exe refresh config.`

   The message `Configuration cache refresh complete` indicates that the refresh operation has finished.

Deleting hierarchy configuration content

Use the delete config option to remove all hierarchy configuration content from Hierarchy Manager, such as names and properties. This allows you to start with new content. Note that this delete process does not remove virtual meters. You need to use Hierarchy Manager to delete virtual meters.

NOTICE

LOSS OF DATA

Performing this operation permanently removes configuration content from the system. Ensure that this operation is necessary before proceeding. This procedure cannot be undone.

Failure to follow these instructions can result in a loss of data.

To delete configuration content:

1. Open a Command Prompt window:

   a. Use Windows Explorer to navigate to:

      `...\WinPM.Net\Applications.`

   b. Click the bin folder and press **Shift + Right-click** and select **Open command window here** on the context menu.

2. Type `HierarchyConfig.exe delete config`. Type `yes` when prompted. The utility deletes all hierarchy configuration content. To confirm the deletion, open Hierarchy Manager. There should be no hierarchy content displayed.

Hierarchy template structure

WinPM.Net includes a number of standard hierarchy templates that provide a structure for modeling the different aspects of your organization. This section describes the structure of a hierarchy template XML file.

Sample and generic hierarchy templates are included with your software. You can use any of these templates if they more closely represent the hierarchy organization you want to implement. However, you need to have a thorough understanding of the structure of a template before

making any changes. Always back up your files before proceeding with any changes. The additional hierarchy templates are located in:

```
...\Applications\HierarchyManager\SampleTemplates
```

> **NOTE**: Any time you change a hierarchy template, you need to import it into Hierarchy Manager before those changes take effect. See Installing a hierarchy template.

Localizing a hierarchy template XML file

You can localize a hierarchy template XML file to display national language text on the tabs and views in Hierarchy Manager, and also wherever hierarchies are enabled in the Web Applications component. For example, localized hierarchy node names will display in the **Source Selector** in the Reports application.

To localize the text you need to edit the displayName attribute that is used throughout a hierarchy template XML file. For example, the display name attribute for `Site` in the hierarchy template file is `displayName="Site"`. Change the text within the quotation marks to the national language equivalent.

The displayName attribute is used in `<Type>`, `<Attribute>`, `<Reference>`, and `<View>` xml elements in a hierarchy template file. (See The XML elements in the template file for descriptions of these elements.) In some cases the same text is used on displayName attributes more than once in the hierarchy template file. Be sure to change every occurrence in the file.

When you complete the edits, follow the process described in Installing a hierarchy template to update the hierarchy template used by Hierarchy Manager.

The hierarchy template

The hierarchy template XML file defines the Node types, attributes, and relationships used in Hierarchy Manager. Each element of the template is defined using information contained within the various XML tags. For example, assume that `Generic 5 Level Template.xml` is imported into Hierarchy Manager. (See the Generic 5 Level Template.)

The initial page in Hierarchy Manager shows the node types and attributes defined in the hierarchy template file. The bottom three node types are configured for devices to be connected to them.

(See Additional notes for considerations related to associating devices to more that one node level in a hierarchy.)

The hierarchy view defined in the template is represented on the Hierarchy Manager **Tree View** page as a physical layout:
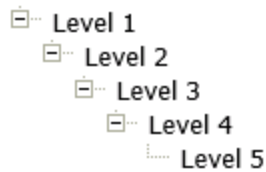
The following table summarizes the major elements in a hierarchy template XML file.

Tag	Description	Example
<Template>	Contains template namespace information	<Template version="1.0" xmlns="uri:application-modules/power/model/template#">
<Types>	Container tag for Type and Attribute elements	<Types> </Types>
<Type>	Defines elements that define characteristics used by the node. Each type represents a tab in Hierarchy Manager.	<Type displayName="level_1".../>
<Attribute>	Defines the characteristics used in each type.	<Attribute displayName="id" dataType="String" uri="uri::application-modules/power/model/attribute#name" />
<Reference>	Used to create relationships between nodes. For a relationship to exist between nodes, each Type element must contain reference information for every other element it is associated with.	<Reference displayName="level_2" isReverse="true" target="uri:application-modules/power/model/type#level_2" uri="uri:application-modules/power/model/relation#1_2" />
<Views>	Contains elements that define how the Node types are presented in different hierarchical views.	<Views> </Views>
<View>	Specifies the node type the view starts at. Contains Relation elements.	<View displayName="Generic Layout" rootNodeTypeUri="uri:application-modules/power/model/type#level_1">
<Relation>	Contained in a View element. Defines the order that the node types are displayed in.	<Relation follow="uri:application-modules/power/model/relation#1_2" reverse="true" />

The XML elements in the template file

This section describes each element in the template XML file.

\<Template\>

The `<Template>` element is the container for all of the XML tags.

```
<Template version="1.0" xmlns="uri:application-
modules/power/model/template#">
```

\<Types\>

This is a containment tag for all `<Type>` elements. Make sure any `<Type>` elements are inside this containment tag.

```
<Types>
    <Type...>
     ...
    </Type>
    ...
</Types>
```

\<Type\>

The `<Type>` tag contains all the information required to define the Node. Each defined type adds a tab in Hierarchy Manager. The tab name is defined by `displayName`. The `<Type>` tag also contains one or more `<Attribute>` and `<Reference>` tags.

```
<Types>
    <Type displayName="level_1" uri="uri:application-
modules/power/model/type#level_1">
        <Attribute... />
        <Attribute... />
        ...
        <Reference... />
        ...
    </Type>
    <Type>
     ...
    </Type>
</Types>
```

To change the tab name in Hierarchy Manager from `level_1` to `new_name`, modify `displayName` in the template file from:

```
<Type displayName="level_1" uri= "uri:application-
modules/power/model/type#level_1">
```

to:

```
<Type displayName="new_name" uri="uri:application-
modules/power/model/type#new_name">
```

\<Attribute\>

The \<Attribute\> tag defines the characteristics of the \<Type\>. It uses dataType to determine the format of the edit dialog that opens in Hierarchy Manager.

You need to define at least one \<Attribute\> element with attribute#name in the uri:

uri:application-modules.com/power/model/attribute#name.

The datatypes on the \<Attribute\> element can be such things as name, floor number, or breaker ratings, depending on the template layout.

- dataType="String" - For short blocks of text (for example, name).

- dataType="LongText" - For long blocks of text (for example, notes, contact information).

- dataType="Number" - For numeric fields (for example, breaker rating in Amps).

- dataType="Boolean" - For checkboxes (for example Rack Circuit).

```
<Types>
    <Type displayName="level_1" uri="uri:application-
modules/power/model/type#level_1">
        <Attribute displayName="id" dataType="String" uri="uri:application-
modules/power/
         model/attribute#name" />
        <Attribute... />
        ...
      <Reference... />
        ...
    </Type>
    <Type>
     ...
    </Type>
</Types>
```

To add an attribute to a node, determine the characteristic you need to capture with that attribute, determine the appropriate dataType, and create the attribute. To define **Name** as one of the characteristics for a type, the Attribute tag is written as:

```
<Attribute displayName="Name" dataType="String" uri="uri:application-
modules/power/model/
  attribute#name" />
```

<Reference>

The <Reference> tag defines the relationship between types. To establish a relationship, each <Type> element needs a <Reference> element, and each <Reference> element needs to refer to the displayName of the related <Type>.

The following example contains references establishing a relationship between the <Type> with displayName="level_1", and the <Type> with displayName="level_2".

Since the types are in a parent-child relationship, the reference in the child needs to contain isReverse="true".

The last portion of the <Reference> element includes a relation name. It is used in the Relation element within the <View> structure to define what appears on the **Tree View** page in Hierarchy Manager.

```
<Types>
    <Type displayName="level_1" uri="uri:application-
modules/power/model/type#level_1">
        <Attribute displayName="id" dataType="String" uri="uri:application-
modules/power
         /model/attribute#name" />
        <Attribute... />
        ...
        <Reference displayName="level_2" uri="uri:applications-
modules/power/model/
         relation#1_2" target="uri:application-
modules/power/model/type#level_2 />
         ...
    </Type>
    <Type displayName="level_2" uri="uri:application-
modules/power/model/type#level_2">
        <Attribute displayName="id" dataType="String" uri="uri:application-
modules/
        power/model/attribute#name" />
        <Attribute... />
        ...
        <Reference displayName="level_1" isReverse="true"
uri="uri:applications-modules/
        power/model/relation#1_2" target="uri:application-
modules/power/model/
        type#level_1 />
        ...
    </Type>
</Types>
```

A physical or logical device can be associated with any type. To configure a type to connect with a device, create a Reference tag similar to the example below. Note that meteredBy is a keyword, and it must be entered exactly as shown to correctly reference a device.

```
<Reference displayName="device" uri="uri:application-
modules/power/model/relation#meteredBy"
  target="uri:application-modules/power/model/type#device" />
```

<Views>

This is a containment tag for all `<View>` elements. Views are the means by which other applications such as Dashboards and Reports discover the set of devices that are aggregated together. All `<View>` elements need to be inside this containment tag.

```
<Types>
    <Type...>
     ...
    </Type>
    <Type...>
     ...
    </Type>
</Types>
<Views>
    <View... >
    </View>
</Views>
```

<View>

The `<View>` tag defines how a hierarchy can be viewed in WinPM.Net. These different views provide different ways that a hierarchy can be displayed, or different ways the device data associated with the hierarchy types can be grouped together.

The view `displayName` (for example **Generic Layout**) is defined here. The name appears on the **Tree View** page of Hierarchy Manager. You can select the view as a data source in the Dashboards and Reports applications.

The `<View>` tag groups together the `<Relation>` elements whose order determines how the view appears.

```
<Types>
    <Type>
     ...
    </Type>
    <Type...>
     ...
    </Type>
</Types>
<Views>
    <View displayName="Generic Layout"
rootNodeTypeUri="uri:application-modules/power/model/
    type#level_1 >
      <Relation... />
      <Relation... />
      ...
    </View>
</Views>
```

<Relation>

The <Relation> tag is used to establish how the different views of the hierarchy are structured. The initial relation is defined in the <Reference> element within a <Type>. Specify the node type to start at (the initial <Relation> entry), then add <Relation> entries to complete the tree.

If a <View> element contains a <Relation> with reverse="true", then the view presented follows the relationship in the reverse order.

```
<Types>
    <Type displayName="level_1" uri="uri:application-
modules/power/model/type#level_1">
        <Attribute displayName="id" dataType="String" uri="uri:application-
modules/power/
        model/attribute#name" />
        ...
        <Reference displayName="level_2" uri="uri:applications-
modules/power/model/
        relation#1_2" target="uri:application-
modules/power/model/type#level_2 />
    </Type>
    <Type displayName="level_2" uri="uri:application-
modules/power/model/type#level_2">
        <Attribute displayName="id" dataType="String"
uri="uri:application-modules/
        power/model/attribute#name" />
        <Attribute... />
        ...
        <Reference displayName="level_1" isReverse="true"
uri="uri:applications-modules/
        power/model/relation#1_2" target="uri:application-
modules/power/model/
        type#level_1 />
    </Type>
    <Type...>
     ...
    </Type>
</Types>
<Views>
    <View displayName="Generic Layout"
rootNodeTypeUri="uri:application-modules/power/model/
    type#level_1 >
        <Relation follow="uri:application-modules/power/model/relation#1_2"
/>
        <Relation follow="uri:application-modules/power/model/relation#2_3"
/>
        <Relation follow="uri:application-modules/power/model/relation#3_4"
/>
        <Relation follow="uri:application-modules/power/model/relation#4_5"
/>
        ...
    </View>
</Views>
```

Additional notes

Considerations when selecting a hierarchy view in a Web-based report

If devices are associated with more than one node in a parent-child relationship in a hierarchy tree, only those devices first encountered in the tree are included in generated reports. All devices lower in the hierarchy tree are ignored in generated reports.

For example, if a hierarchy tree is structured as:

Site 1

Building 1 (including associated devices)

Area 1 (including associated devices)

and you select **Building 1** on the **Views** page of the **Source Selector** in a report, the generated report includes data for the devices associated with **Building 1**, but not for devices in the lower node in the tree – **Area 1**.

Similarly, if you select **Site 1**, which does not include a device association in the hierarchy, the generated report includes data for the devices associated with the **Building 1** level only, the level in the hierarchy where devices are first encountered.

Using comment tags

If you do not require all the levels provided in the example hierarchy template XML file, you can comment out the types you do not need, from the bottom type up. Commenting out the type, rather than just deleting the type preserves the structure of the template file and allows you to reinstate the type with a minimum of effort. Enclose the `<Type>` element within the start comment tag `<!--` and the end comment tag `-->` to comment it out. For example:

```
<!-- <Type... >
...
</Type> -->
```

Remove the comment tags to restore the type.

If you want to remove one or more view elements defined in a template, add comment tags around the applicable `<View>` tags. Remove the comment tags to restore the View elements.

When you import the hierarchy template XML file, the Hierarchy Configuration utility performs a validation test on the file. If the file validation is not successful, the utility does not import the file and provides information indicating where the validation process stopped. Check for any discrepancies in the file, save the changes and reimport the file.

Generic 5 Level Template

```
<Template version="1.0" xmlns="uri:application-modules/power/model/template#">

<!-- This is a template for a simple 5 level hierarchy. All the relations and nodes have been pre-configured. To customize for your purposes, search and replace "level_1" with the name of your top-level type (e.g. Site),
...
"level_5" with the name of your lowest-level type (e.g. Room)
With this template you can attach "devices" to the lower 3 levels.
-->

<Types>

<Type displayName="level_1" uri="uri:application-modules/power/model/type#level_1">
<!-- every Type must have attribute#name - change displayName as needed -->
<Attribute displayName="id" dataType="String" uri="uri:application-modules/power/model/attribute#name" />
<!-- use these elements to add attributes to nodes -->
<Attribute displayName="string" dataType="String" uri="uri:application-modules/power/model/attribute#str" />
<Attribute displayName="notes" dataType="LongText" uri="uri:application-modules/power/model/attribute#note" />
<Attribute displayName="number" dataType="Number" uri="uri:application-modules/power/model/attribute#num" />
<Attribute displayName="boolean" dataType="Boolean" uri="uri:application-modules/power/model/attribute#bool" />

<!-- this is how to create one half of the relationship between two nodes, you need a similar reference on the other end of the relationship -->

<Reference displayName="level_2" uri="uri:application-modules/power/model/relation#1_2" target="uri:application-modules/power/model/type#level_2" />
</Type>

<Type displayName="level_2" uri="uri:application-modules/power/model/type#level_2">
<Attribute displayName="id" dataType="String" uri="uri:application-modules/power/model/attribute#name" />
<!-- this is the other end of the relationship marked above - isReverse is opposite, relation#... must be equal -->
<Reference displayName="level_1" isReverse="true" uri="uri:application-modules/power/model/relation#1_2" target="uri:application-modules/power/model/type#level_1" />
<Reference displayName="level_3" uri="uri:application-modules/power/model/relation#2_3" target="uri:application-modules/power/model/type#level_3" />
</Type>

<Type displayName="level_3" uri="uri:application-modules/power/model/type#level_3">
<Attribute displayName="id" dataType="String" uri="uri:application-modules/power/model/attribute#name" />
<Reference displayName="level_2" isReverse="true" uri="uri:application-modules/power/model/relation#2_3" target="uri:application-modules/power/model/type#level_2" />
<Reference displayName="level_4" uri="uri:application-modules/power/model/relation#3_4" target="uri:application-modules/power/model/type#level_4" />
<Reference displayName="device" uri="uri:application-modules/power/model/relation#meteredBy" target="uri:application-modules/power/model/type#device" />
</Type>

<Type displayName="level_4" uri="uri:application-modules/power/model/type#level_4">
<Attribute displayName="id" dataType="String" uri="uri:application-modules/power/model/attribute#name" />
<Reference displayName="level_3" isReverse="true" uri="uri:application-modules/power/model/relation#3_4" target="uri:application-modules/power/model/type#level_3" />
<Reference displayName="level_5" uri="uri:application-modules/power/model/relation#4_5" target="uri:application-modules/power/model/type#level_5" />
<Reference displayName="device" uri="uri:application-modules/power/model/relation#meteredBy" target="uri:application-modules/power/model/type#device" />

</Type>

<Type displayName="level_5" uri="uri:application-modules/power/model/type#level_5">
<Attribute displayName="id" dataType="String" uri="uri:application-modules/power/model/attribute#name" />
<Reference displayName="level_4" isReverse="true" uri="uri:application-modules/power/model/relation#4_5" target="uri:application-modules/power/model/type#level_4" />
```

```
<!-- this is how to connect up a logical/physical device to provide metered data to a node this can appear anywhere in the
hierarchy e.g. main meter for building, metering points for specific loads-->
<Reference displayName="device" uri="uri:application-modules/power/model/relation#meteredBy" target="uri:application-
modules/power/model/type#device" />
</Type>

</Types>

<Views>

<!-- View displayName shows up in the drop-down of Reports/Gadgets -->
<!-- specify the node type to start at, then add the relations to follow -->
<View displayName="Generic Layout" rootNodeTypeUri="uri:application-modules/power/model/type#level_1">
<Relation follow="uri:application-modules/power/model/relation#1_2" />
<Relation follow="uri:application-modules/power/model/relation#2_3" />
<Relation follow="uri:application-modules/power/model/relation#3_4" />
<Relation follow="uri:application-modules/power/model/relation#4_5" />
</View>

<!-- do not need to add the "meteredBy" relation in the View if a "meteredBy" relation has a mapped device no matter what level
of the hierarchy then the aggregation will return the value from the mapped device -->
</Views>
</Template>
```

Chapter 8: Downstream Device Assistant

Downstream Device Assistant lets you manage downstream devices. With this tool, you can rename downstream sources, change which measurement is associated with specific data from a source, and apply these changes to previously recorded data in ION_Data (historical database). You can also display and manage all the physical sources (devices) in your network.

Downstream Device Assistant is an advanced application and should only be used by users with advanced technical skills and a clear understanding of how data is interpreted and stored in the WinPM.Net databases.

## ⚠ CAUTION

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.

Failure to follow these instructions can result in injury or equipment damage.

Detecting downstream devices

A downstream device is one whose data is logged by a remote data recorder, for example:

- A Modbus slave device whose output registers are logged by a device with Modbus Mastering capability (see illustration below).
- Any of various multiple sources in which the same measurement is being logged on a single device, for example, an RTU that is collecting kWh pulses from multiple sources or a single branch circuit monitor that is collecting current readings on multiple circuits.
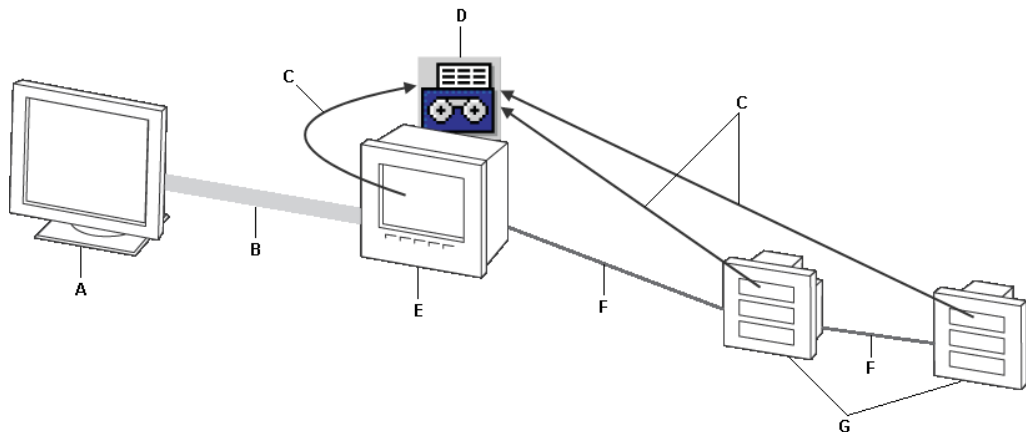
| A | WinPM.Net server | B | Ethernet | C | kW | D | Data Recorder |

E	Physical source (Modbus Master)	F	RS-485	G	Downstream Devices (Modbus Slaves)

WinPM.Net automatically detects downstream devices. The Log Inserter component does this by looking for devices (called "physical sources") that record or host data coming from various downstream devices (for example, an ACCESS meter that utilizes a Modbus Import module to pass through data originating from a "downstream" Modbus device).

Log Inserter resolves the data from each of these downstream devices to the appropriate measurement (for example, the label "Vll ab" resolves to the measurement "Voltage Phases AB"). For data that cannot be resolved, Log Inserter creates measurements whose names are based on the ION register labels.

Log Inserter uses a logical naming scheme to assign names to the downstream devices. For example, "Modbus.34" is the name assigned to the downstream Modbus device with a slave address "34".

Downstream Device Assistant basics

Use Downstream Device Assistant to edit the names assigned to downstream devices or to assign the correct measurement label for a register that has an incorrect measurement mapping.

IMPORTANT: Before using the Downstream Device Assistant, make sure you save a copy of the ION_Network and ION_Data databases, in case you need to revert to the original configuration of these databases.

Starting the Downstream Device Assistant

1. Run the program named "DownstreamDevices.DownstreamAssistant.exe" in the ...\WinPM.Net\system\bin folder.

2. Log in using your WinPM.Net credentials.

   A message displays to remind you to back up your ION_Network and ION_Data databases. Click **OK** to continue.

3. Downstream Device Assistant opens in Basic view or Advanced view, depending on the last view it was in before it was closed. The first time Downstream Device Assistant is started, it opens in Basic view.

Basic view versus Advanced view

The view determines what columns are available as well as whether an action is global (applied to all rows containing a parameter) or specific to a row.

Columns descriptions

Columns in Both Basic and Advanced View	
Column Name	**Description**
IsDownstream[1]	The **IsDownstream** column identifies which devices in the **Assigned Source** column are downstream devices. If **IsDownstream** is selected, it means that the device identified in the **Assigned Source** column is downstream to a physical device, whose name is indicated in the **Physical Source** column. If **IsDownstream** is cleared, this indicates that the device is a physical source. Note also that a physical source has the same **Assigned Source** name and **Physical Source** name, by default.
Assigned Source	This is the default name that Log Inserter assigned to a downstream or physical device. For example, "Modbus.109:MainFeed.9610" is the default name assigned for the downstream Modbus device with unit ID "109", whose measurements are being logged to a data recorder on a Modbus Master device named "9610" that is part of the "MainFeed" site (group).
Updated Assigned Source	This text field allows you change the current name under **Assigned Source**. In Basic view, all rows that contain the same **Assigned Source** name are automatically updated to the new name when you enter it in the **Updated Assigned Source** column. In Advanced view, you can select one or more rows and rename the source using the **Updated Assigned Source** column, while leaving the other rows unchanged. See Renaming the assigned source for more details.
Physical Source[2]	This identifies the name of the physical device, for example a Modbus Master device, that is performing the data logging for a downstream device.
Measurement	This column displays either the ION label (for example, "Vln avg") or the descriptive name of the measurement (for example, "Average Voltage Line-to-Neutral"), depending on the setting in the Options dialog; see Downstream Device Assistant options.
Additional Columns in Advanced View	
Updated Measurement	This lets you redefine what is currently assigned in the Measurement column. Click the cell to display the list of measurements to choose from. Depending on the setting in the Options dialog, you can select from a list containing only the most common measurement definitions or from a list of all available measurements; see Downstream Device Assistant options.

Columns in Both Basic and Advanced View	
Handle	This column displays a measurement's register handle either in hexadecimal format or decimal format, depending on the setting in the Options dialog; see Downstream Device Assistant options.
Entered By	This identifies which user made a change to the source-measurement definition.
Update Historical Database	The **Update Historical Database** check box is automatically selected after entering changes in the **Updated Assigned Source** or **Updated Measurement** cell. This means that all historical database entries that contain the old **Assigned Source** name or **Measurement** will be updated with the new name or measurement. If you want to leave the existing data in the database unchanged, clear the **Update Historical Database** check box before saving the changes.
Manual (see the Note below)	By default, Log Inserter automatically associates the proper measurement to the appropriate piece of data in the data log. Normally, the **Manual** check box should remain cleared. However, if you want to specifically change the measurement or register handle associated with a particular address definition, select the **Manual** check box for that measurement. This manually overrides Log Inserter so that it uses this new address mapping from now on (i.e., Log Inserter does not attempt to resolve that address ever again).

[1] Do not change the value of "IsDownstream" when making changes to source/measurement definitions, as it is automatically set by the Downstream Device Assistant

[2] Do not use the Physical Source column to modify the existing name of a valid physical device, as Log Inserter uses this name to perform proper data logging.

NOTE: It is **not** recommended that you select the Manual check box. This reserves the ION register for which Manual is checked for the selected measurement. This mapping remains in place even if the device is reconfigured such that this register is used in an entirely different context. This may cause unexpected problems in the ION_Data database. It should be used only if you are certain that the configuration you have chosen will not change.

Basic view

To switch to Basic view, click **View > Options**. In the **Advanced** section of the **Options** dialog, clear the **enable advanced features** check box. Click **OK**.

In Basic view, when you enter a new name in the **Updated Assigned Source** column, all rows that contain the same **Assigned Source** name are automatically filled in with the new name. Existing data in the historical database (ION_Data) is also automatically updated to reflect the change.

Advanced view

To switch to Advanced view, click **View > Options**. In the **Advanced** section of the Options dialog, select the **enable advanced features** check box. Click **OK**.

In addition to the columns displayed in Basic view, Advanced view displays all the other columns for editing individual source-measurement definitions (i.e., Updated Measurement, Handle, Entered By, Update Historical Database, and Manual). In Advanced view, you can:

- Select and make changes to a specific row without affecting the other rows. For example, you may want to change the source name for one particular measurement while leaving the original source name unchanged for all the other measurements.

- Select a different measurement for one that is incorrectly mapped to a particular register handle.

- Select whether or not to update historical data for a changed source name or measurement definition, by selecting or clearing the **Update Historical Data** check box, respectively. Changing a name or definition automatically selects this check box.

- Select whether or not to allow Log Inserter to resolve the register handle to a particular source measurement, by clearing or selecting the **Manual** check box, respectively. The default setting is cleared (i.e., Log Inserter automatically resolves the register handle to the source measurement definition). If you want to force the change so Log Inserter does not resolve the handle to the new measurement, select the **Manual** check box.

Displaying physical sources

The default view for Downstream Device Assistant displays only the downstream devices. To display the physical sources as well:

1. Click **View > Select Physical Sources**.

2. Select the check box beside the physical sources you want to display. Select the **Select/Clear All** check box to select all physical sources or clear it to select none. Click **OK**.

NOTE: The Physical Sources are not displayed if you restart Downstream Device Assistant. By default, only downstream devices are displayed when Downstream Device Assistant is started.

Sorting data

Click a column heading to sort the row entries in ascending or descending alphanumeric order, according to that column.

Downstream Device Assistant options

Click **View > Options** to open the **Options** dialog. The dialog contains Measurements, Historical Data, and Advanced options.

Certain options such as "show all measurements" and the "Handles" display options are disabled in Basic View. To enable them, switch to Advanced View first (see Advanced).

Measurements

This section allows you to select whether to display the default ION label (for example, "Vln avg") in the Measurement column or to display the full descriptive name of the measurement (for example, "Average Voltage Line-to-Neutral").

In Advanced view only, you can also select whether to display all available measurements or only the most common measurements in the list when picking a measurement in the Updated Measurement column.

Displaying ION labels versus measurement names

To change the setting:

1. Click **View > Options**.

2. In the Measurements section, select **display ION labels** to display the ION label or **display measurement names** to display the full name of the measurement.

NOTE: In some situations, if "display ION labels" is selected, the displayed label may not match the label on your meter (for example, if a custom label was used). Instead, it displays the known default label for that particular measurement.

Updated Measurement column display

To change the setting:

1. Click **View > Options** and ensure Advanced is selected.

2. In the Measurements section, select **show all measurements** to display all available measurements in the **Updated Measurement** column. To display only the most common measurements, clear **show all measurements**.

Historical data

If the **show connection dialog** check box is cleared, the ION_Data database is automatically set as the "Default Connection" (for historical data).

If you want to select a different database, select the **show connection** dialog. With this setting enabled, the database connection dialog is displayed every time you save your changes. When the dialog opens, select **Custom Connection** and specify your database connection details.

Advanced

The **enable advanced features** check box sets the view to Basic View if it is cleared or Advanced View if it is selected.

Handle

Available in Advanced View only, this option specifies whether register handles are displayed as the hexadecimal value (with the "0x" prefix) or as the decimal value.

To change the setting:

1. Click **View > Options** and ensure the Advanced is selected.

2. In the Handles section, select **display hex** to display the register handle in hexadecimal format or **display decimal** to display it in decimal format.

If you want to manually resolve a register handle to a specific measurement, edit the value in the **Handle** cell, then select the **Manual** check box before saving the changes.

Downstream Device Assistant functions

This section describes different functions you can perform using the Downstream Device Assistant.

⚠ CAUTION

INACCURATE DATA RESULTS

- Do not incorrectly configure the software or the devices.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.

Failure to follow these instructions can result in injury or equipment damage.

Renaming the assigned source

One of the primary reasons for using the Downstream Device Assistant is to change the default name assigned to a downstream device by the Log Inserter. For instance, instead of "Modbus.34", you may want to rename that device to something more meaningful such as "SM_Bldg2_Boiler7".

Renaming an **Assigned Source** typically only applies to downstream devices.

Renaming the assigned source for all its measurements

1. Switch to Basic view.

2. Find the source you want to rename. Enter the new name in the **Updated Assigned Source** column. All rows with measurements associated with the renamed source are automatically updated.

   If you do not want to assign the existing data in the database to the new source name, switch to Advanced View first, then clear the Update Historical Database check box for all appropriate rows.

3. Click **File > Save**.

4. A dialog opens showing a summary of the changes. Click **Yes** to continue. or **No** to cancel.

5. When the operation completes, click **OK**, then **Close**.

Renaming the assigned source for one or more measurements

1. Switch to Advanced View.

2. Select the row(s) you want to rename. To select a row, click the area just to the left of the first column. The row is highlighted to indicate that it is selected.

3. Click **Edit > Rename Source**.

4. In the **Source Rename** box, type the new name for the downstream source. Click **OK**.

> **NOTE**: By default, the Update Historical Database check box is selected for all affected rows — this applies the change to existing data in the database. To make the change only to data going into the database from this point forward, clear the Update Historical Database check box for the affected rows.

5. Click **File > Save**. A dialog open showing a summary of the changes.

6. Click **Yes** to continue or **No** to cancel.

7. When the operation has completed, click **OK**, then click **Close**.

Cancelling changes

1. To undo all unsaved changes:
   - Click **Edit > Reset**, or
   - Click **View > Refresh**.

2. Click **Yes** to confirm.

Changing the measurement mapping

Log Inserter automatically associates logged data with a measurement by searching for a mapping between the label and the measurement. However, in cases where no mapping exists, Log Inserter creates a "measurement" that is based on the register label. To manually change this measurement mapping:

1. Switch to Advanced View.

2. Click the **Updated Measurement** cell that you want to change, then select the new measurement you want to map from the list.

3. Select the **Manual** check box.

4. By default, the **Update Historical Database** check box is selected — this applies the change to existing data in the database (if, for example, the source and measurement mapping you are adding already exists). To make the change only to data going into the database from this point forward, clear the **Update Historical Database** check box.

5. Click **File > Save**. A dialog opens showing a summary of the changes. Click **Yes**.

6. If prompted, select which database to update. Click **OK**.

Adding a new downstream device mapping

To add a new address mapping that associates the source and measurement to a specific logged quantity in the database:

1. Scroll down to the end of the table and click the last row (the one with the asterisk beside it).

2. Fill in the **Updated Assigned Source** and **Physical Source** columns. If the source is a physical device (i.e., not downstream), the same name must be entered exactly in both columns.

3. Click the **Updated Measurement** cell, then select the measurement from the list.

4. In the **Handle** cell, type the register handle for this measurement.

5. By default, the **Update Historical Database** check box is selected — this applies the change to existing data in the database (if, for example the source and measurement mapping you are adding already exists). To make the change only to data going into the database from this point forward, clear the **Update Historical Database** check box.

6. Click **File > Save**. A dialog opens showing a summary of the changes. Click **Yes**.

7. If prompted, select which database to update. Click **OK**.

Updating the historical data at a later time

When you save your changes using the Downstream Device Assistant, a record of changes to the source-measurement definitions is saved in the folder:

...\WinPM.Net\config\Diagnostic\Downstream Device Assistant\

with the following filename:

"Downstream Device DefinitionsYYYY-MM-DD_hh.mm.ss.csv"

where "YYYY-MM-DD_hh.mm.ss" means current YEAR-MONTH-DAY_hour.minute.second.

If you cleared the **Update Historical Database** check box when you saved your changes the first time, you can update the existing data in the database at a later time, through the use of this csv file.

How to use the Downstream Device Assistant .csv file

To update the existing data in the database with the source-measurement changes contained in the csv file:

1. Click **Edit > Historical Data**. The Update Historical Data window appears.

2. Click **File > Import CSV**.

3. Downstream Device Assistant opens the folder that contains the csv files. Select the Downstream Device Definitions file you want to use, then click **Open**.

4. Click **File > Update Historical Data**.

5. The Historical Data Update prompt displays how many source-measurement pairs will be updated. Click **Yes**.

6. Click **Yes** at the Historical Data Update prompt.

7. After the operation has completed, click **OK**, then **Close**.

You can update only one ION database at a time. To select a different database, first make sure the **show connection dialog** check box is selected under **View > Options**. You can then use the database connection dialog to specify which database you want to update.

NOTE: If you update address definitions that affect multiple historical databases, only the first historical database you select is automatically updated. To update the remaining historical databases, you must import the address definitions to each one, using **Edit > Update Historical Data**.

Activity log

The saved changes in the Downstream Device Assistant are stored in a log file named "Downstream Device Historical UpdatesYYYY-MM-DD_hh.mm.ss.txt", that contains a description of the activities carried out when Downstream Device Assistant was updating the source-measurement definitions.

Downstream Device Assistant commands

The following summarizes the commands available in the Downstream Device Assistant:

- **File > Save**: Saves your changes. A copy of the changes are saved for future reference, in a comma-separated value (.csv) data file in the ...\config\Diagnostic\Downstream Device Assistant\ folder.

- **File > Exit**: Closes the Downstream Device Assistant.

- **Edit > Reset**: Clears the modifications made since the last save.

- **Edit > Delete**: Deletes the selected rows.

- **Edit > Rename Source**: Lets you change the name of an **Assigned Source**. See Renaming the assigned source.

- **Edit > Update Historical Data**: Opens the "Update Historical Data" window that allows you to open a previously-saved source-measurement definition data file (with a .csv file extension), for the purpose of updating existing data in the database. You can also manually add, edit or delete data entries. See Updating the historical data at a later time.

- **View > Refresh**: Loads the current downstream definitions from the database. Functionally the same as **Edit > Reset**, as this command clears the modifications made since the last save.

- **View > Options**: Displays the **Options** dialog where you can change the settings that control what level of detail is displayed for the source-measurement definitions.

- **View > Select Physical Sources**: Displays the **Select Physical Sources** dialog where you can select which physical devices you want displayed in the Downstream Device Assistant.

Chapter 9: OPC Server Support

Introduction to OPC

OPC is a set of open standards for connectivity and interoperability of industrial automation and the enterprise system. OPC provides a bridge between Windows based applications and process control hardware, thereby eliminating the need for proprietary or custom interfaces and drivers for the various data types and sources residing in your corporate information network. Having information readily available in a universally-recognized format can cut costs, speed up development and increase operations efficiency. This is especially true when many diverse software applications, hardware, and operating systems exist in the corporate and operations networks.

RECOMMENDATION: Use an OPC tunneling application for OPC data exchange over a network

---

⚠ WARNING

INACCURATE DATA RESULTS

- Do not incorrectly configure the OPC Server.
- Do not base your maintenance or service actions solely on messages and information displayed by the software.
- Do not rely solely on software messages and reports to determine if the system is functioning correctly or meeting all applicable standards and requirements.
- Consider the implications of unanticipated transmission delays or failures of communications links.

Failure to follow these instructions can result in death, serious injury, equipment damage, or permanent loss of data.

OPC is supported through the implementation of WinPM.NetReal Time Data Service, a .NET-based system that takes the traditional "register handle-based" ION data and transforms it into structured "measurement-based" data that complies with open standards such as OPC. WinPM.Net Real Time Data Service facilitates the translation and organization of data in this new measurement classification system.

WinPM.Net supports OPC server and OPC client functionality. The OPC server translates ION data into OPC data, for exporting and viewing in other third-party OPC client systems. The OPC client, on the other hand, takes OPC standardized measurements from third-party systems and translates them into a data format that WinPM.Net can use.

This appendix provides basic configuration and operation instructions for the OPC server component of WinPM.Net.

NOTE: The OPC client is available in all WinPM.Net installations. However, due to the number of different third-party OPC-compliant servers in the industry, as well as different methods and syntax for connecting to these servers and accessing their data, it is highly recommended that you contact Technical Support for assistance in configuring the OPC client.

Optional OPC server license

WinPM.Net can be ordered with or without the optional OPC server license. You can also order the OPC server license at a later date. If you purchased your OPC server license after you have installed WinPM.Net, enable the OPC server functionality simply by entering your new product key in Management Console (under **Tools > License Manager**).

OPC server type

The WinPM.Net OPC server complies with the "OPC Data Access Custom Interface Standard Version 2.05A". The class of data that the current release of OPC Data Access server is able to provide is Read-Only data (i.e., control functions are not currently supported).

OPC measurement support

OPC measurement support for WinPM.Net allows you to easily translate data from ACCESS devices and compatible Modbus devices into OPC-compliant measurement data formats, then make this measurement data available to OPC clients through the OPC server.

Management Console has a command that lets you export default measurements from compatible devices to the OPC server. In addition, Designer and Device Type Editor user interfaces let you view and select which OPC measurement data you want the OPC server to expose to OPC clients.

Creating/updating default OPC mapping

WinPM.Net can create the default ION-to-OPC data mapping based on the devices contained in the ION_Network database. This default mapping translates the most commonly-used WinPM.Net data to OPC data and exports the data to the OPC server.

1. Launch Management Console.

2. Click **Tools > System > Update OPC Server**.

3. Click **Yes** to create/update the OPC server.

   A summary of the configuration details is displayed.

4. Click **OK**.

> **NOTE**: If you add a new device to the ION_Network database at a later date, you must repeat the above procedure in order to map the new source device's data to the OPC server.

OPC Server Assistant

The OPC Server Assistant is a mechanism that lets you select which OPC measurements to expose to the OPC server. Custom labels are not mapped to OPC server by default. They need to be manually exported to OPC server.

OPC Server Assistant is accessible through Designer.
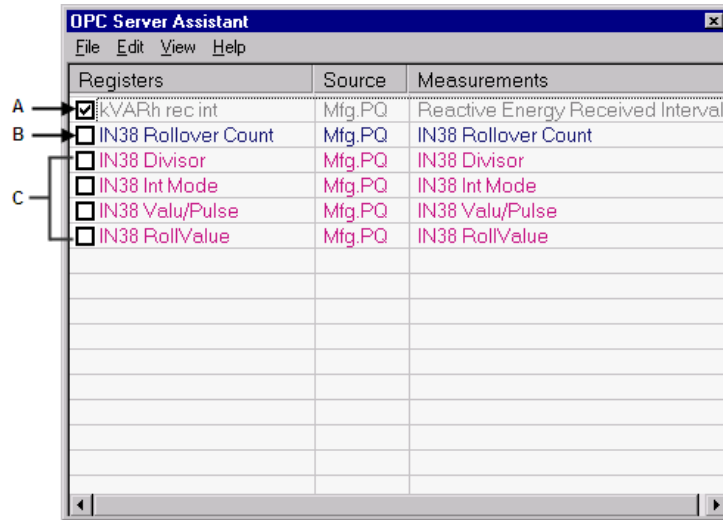
A	Unavailable registers (Grayed-out)	B	Output Register	C	Setup Registers

Exporting OPC measurements through Designer

If you customized an ACCESS device's default templates (for example, custom labels), those measurements are not exposed to the OPC server by default. To manually export those measurements to OPC server:

1. Launch Designer and open the ACCESS device node.

2. Navigate to the ION module that contains the register that you want to expose to OPC server. Right-click to display the module setup dialog.

3. Click **OPC** to open the OPC Server Assistant.

4. Select (check) the box beside each item you want to expose to the OPC server. Clear (uncheck) the box beside each item you do not want to expose to the OPC server.

   NOTE: Grayed-out items (such as "PF sign a" in the Power Meter module) cannot be selected or changed. Those registers cannot be removed using OPC Server Assistant.

5. Click **File > Export**. Click **Yes** to save your changes.

Description of commands

This section describes the commands available in the OPC Server Assistant.

Command	Result
File > Export	Saves your configuration changes. Items that are selected (checked) in the OPC Server Assistant are exposed to the OPC server, while cleared (unchecked) items are not.
File > Exit	Closes the OPC Server Assistant.
Edit > Reset values	Discards your changes and reverts to the last saved configuration (for the items that are exported to the OPC server).

Command	Result
Edit > Select All	Selects (checks) all registers.
Edit > Clear All	Clears (unchecks) all registers that are not grayed out.
View > Setup Registers	Displays the setup registers, in addition to the output registers, for the module.
View > Measurements	Displays a column containing the measurement names associated with the registers.

Viewing OPC data

A third-party OPC client can connect to the WinPM.Net OPC server using a valid connection address, which consists of the network node or host machine name where the OPC server is running and the OPC program ID (ION.OpcDaServer). For example:

```
opcda://WORKSTATION3/ION.OpcDaServer
```

NOTE: Syntax use varies across different OPC clients. Refer to your OPC client documentation for details.

Chapter 10: Time synchronization for SIEMENS 9xxx series meters

You can synchronize the internal clocks of all networked SIEMENS 9xxx series meters and devices through the Management Console component of WinPM.Net. When they are synchronized, all data logs have time stamps that are relative to a uniform time base.

> **NOTE**: This information is specific to SIEMENS 9xxx series devices. It is not applicable to devices that are not based on ION architecture.

Time synchronization signals are broadcast periodically over the network; each meter continually assesses its ability to remain synchronized with the incoming broadcasts. Over a brief period, each meter learns how its internal timing differs from that of the broadcast source and adjusts its timekeeping to compensate. Very accurate time synchronization is achieved with this method.

Meters at modem sites are synchronized each time they are connected. The longer the duration between connections, the larger the error in time synchronization. In the extreme case, this can result in missing or duplicated logs. If this occurs, you can increase the frequency of connections through the WinPM.Net software connection.

> **NOTE**: This information assumes that you are using the time synchronization function in WinPM.Net. Although other methods of time synchronization are available, such as with ION Setup, a Network Time Protocol (NTP) server, a Global Positioning System (GPS) receiver, or 3rd party protocols, it is important that you use only one method of time synchronization on each network. If you use multiple methods, the timestamps will differ for the sites and devices that are using separate time synchronizing methods.

Before you configure time synchronization on your network, you should familiarize yourself with the Clock module settings. Once you have done this, you can decide which synchronization method you want to use.

Enabling and configuring time synchronization

Time synchronization in WinPM.Net is initially disabled and needs to be enabled and configured through Management Console. The communications port and protocol used for communications between the software and the networked SIEMENS 9xxx series devices is automatically used to send time synchronization signals to all connected SIEMENS 9xxx series devices. The software sends a time sync packet and the time is set when the packet is received.

When reviewing time synchronization messages in the system log, remember that the time in the message is not the time to which the meter was time synced, but rather it is the time the message was posted to the system log. The message is posted after the meter is time synced.

Note that ION time synchronization only uses the UTC setting; the LOCAL setting cannot be used.

Time synchronization values are set when sites or Ethernet devices are defined in a WinPM.Net network. Enable time synchronization or set custom intervals for supported devices in any site through Management Console as follows:

1. Start Management Console.

2. Select **Sites** or **Devices** on the System Setup pane:

   - Select **Sites** if you want to customize a particular serial, modem, or Ethernet Gateway
     site.

   - Select **Devices** if you want to customize an individual Ethernet device.

3. Right-click the device or site and select **Configure Device** or **Configure Site** to open the
   related configuration dialog.

4. Right-click inside the dialog and select **Advanced Properties**.

5. Configure the **Time Synch ION Enabled** or **Time Synch Ethernet Enabled**, and **Time
   Synch Interval Ethernet** fields as required for your system.

6. Click **OK** to save you changes.

   The default time synchronization interval of 3600 seconds (displayed in milliseconds) is
   acceptable in most software installations.

NOTE: You need appropriate permissions to configure meters on your network. Refer to the
WinPM.Net System Security technical note for details on software and meter security.

Time synchronization blackout

Time synchronization blackout is defined as a period when time synchronization cannot occur.
Utilities often record power usage at regular, predetermined intervals. For example, the utility may
schedule a recording every five minutes during an hour period (that is, 5, 10, 15, 20,..., 55, 60). If a
time sync moves the meter clock forward, the meter may miss one of the recording intervals. If a
time sync moves the meter clock backwards, the meter may get two records with the same
timestamp. The time synchronization blackout feature seeks to protect the time before and after
these recording intervals by not sending out any time sync signals at those times.

WinPM.Net has time synchronization blackouts enabled automatically even though time syncing
is disabled by default in WinPM.Net. The blackout intervals are every five minutes as follows: 0, 5,
10, 15, 20, 25, 30, 35, 40, 45, 50, 55. Each blackout is 2 minutes and 30 seconds in duration. This
duration is distributed evenly before and after the interval. The diagram below illustrates the time
synchronization blackout feature in WinPM.Net.
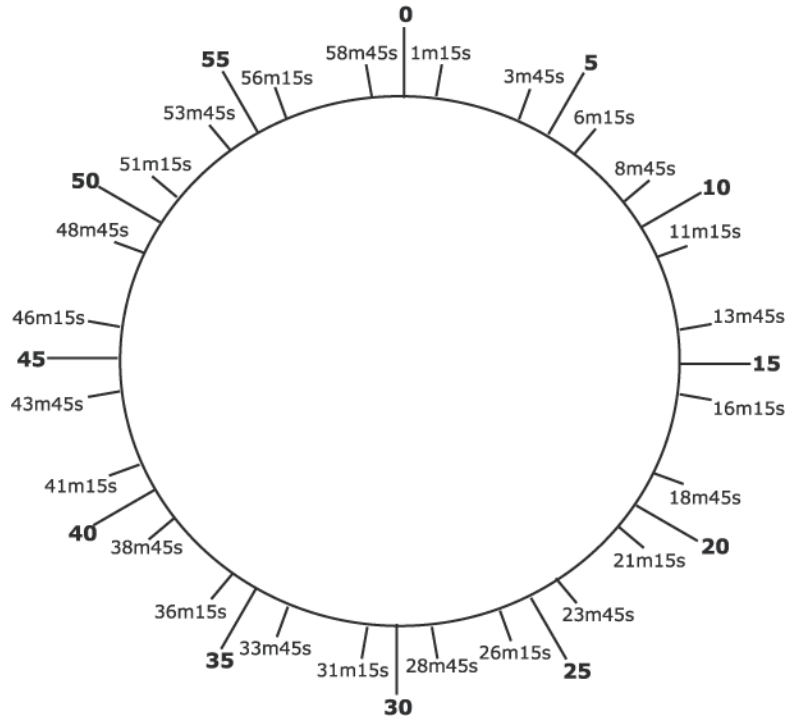
If a regularly scheduled time sync is not sent because of a blackout interval, the software continues trying to send the time synchronization signal every 15 seconds until the blackout period expires and the time sync is sent.

Changing default blackout settings

To change the default settings for the interval or the blackout duration, you must edit the registry of the computer where WinPM.Net is installed. Changing the default registry settings should be performed only by personnel with a thorough knowledge of their computer's operating system and registry entries.

NOTICE

IRREVERSIBLE OPERATING SYSTEM DAMAGE OR DATA CORRUPTION

Before making any changes, back up your Windows Registry in a network folder or other remote location.

Failure to follow these instructions can result in irreparable damage to the operating system of the computer and all existing data.

NOTE: Registry edits must be performed only by qualified and experienced personnel.

There are two registry entries that control the time-synchronization blackout behavior.

Create the following registry `REG_DWORD` values under the `HKEY_LOCAL_ MACHINE\Software\Wow6432Node\Siemens\WinPM.Net\<version_number>` key.

TimesyncBlackoutInterval_mins

Default value: 5

This entry is specified in minutes. It must have a value greater than 1 in order for time-synchronization blackouts to occur. A value of 60 or greater configures blackouts to occur once an hour.

TimesyncBlackoutDuration_secs

Default value: 150 (2 minutes 30 seconds in seconds)

This entry is specified in seconds. A value of 0 (zero) disables the blackout feature. This entry cannot have a value greater than or equal to 3600 (one hour).

NOTE: If you want to disable the blackout feature but keep time synchronization enabled, set the TimesyncBlackoutDuration_secs register to 0.

Do not use the following settings. They disable both the blackout feature as well as time synchronization in general.

- TimesyncBlackoutInterval_mins < 1

- TimesyncBlackoutDuration_secs >= 3600 (1 hour)

- TimesyncBlackoutDuration_secs (the value converted to minutes) >= TimesyncBlackoutInterval_mins

To disable time synchronization for a particular device, use Management Console. Refer to Enabling and configuring time synchronization.

Time-synchronization blackout considerations

- Remember to specify TimesyncBlackoutInterval_mins in minutes and TimesyncBlackoutDuration_secs in seconds.

- If your blackout values conflict or are out of range, an error is sent to the system log database. If time synchronization signals are not occurring, check the system log to see if there's an error message about your blackout values.

Chapter 11: System maintenance and Disaster recovery

This section provides recommendation related to system maintenance and disaster recovery.

Use the links in the following table to find the content you are looking for:

Topic	Content
System maintenance	Provides information on database maintenance and recommended actions.
Disaster recovery strategy	Provides information on disaster recovery and recommended strategy.

System maintenance

Once a WinPM.Net system is installed and commissioned, it must be maintained on a regular basis. Without periodic, proactive system maintenance, system performance degrades over time and the system operates with poor performance (timeouts and sluggish response) and eventually becomes unusable.

This section provides guidelines and recommendations for creating a maintenance schedule. Proper adherence to the maintenance guide ensures a steady-state operation of a WinPM.Net system.

This section does not detail troubleshooting system issues, but identifies the steps to develop a maintenance plan that should allow for a commissioned system to continuously run at optimal performance.

A maintenance plan keeps your WinPM.Net system healthy. A comprehensive maintenance plan starts in the system planning stage before commissioning. The plan should include frequent monitoring and system reviews, and maintenance tasks to ensure the system is operating under healthy conditions.

The following sections provide recommendations for developing your maintenance plan:

- Designing for maintenance

- Database maintenance

- System health review

Designing for maintenance

Before installing and deploying a system, it is important to plan and design the IT architecture in support of maintenance and backup activities. Maintenance and backup activities often require additional resources (CPU, RAM and hard drive space) to perform and complete the activity. The recommendations are as follows:

- System sizing

  Size a system for the final state (for example: maximum devices, maximum logging parameters, etc) to ensure that the system runs in a healthy state for a longer time. See Introduction chapter of Install Guide for recommended system sizing.

- Storage space allocation

  Make sure there is enough hard drive space to perform database maintenance, archive and trim operations. See Storage Performance and Availability topic of Install Guide for recommended hard drive sizing.

- Disk I/O performance

  Disk read/write operations impacts system performance. Plan to have different storage devices installed for the system files, the database files, and the Microsoft Message Queuing (MSMQ) files. See Choosing Data Storage topic of Install Guide for recommended storage information.

- Pre-size databases

  By default, ION_Data database is configured with a 10% auto-growth parameter. This growth operation can lead to a fragmented database and hard drive and can impact database performance over time. To minimize the use of the auto-growth feature, pre-allocate hard drive space for the ION_Data database during system commissioning, including systems running SQL Server Express. See Database growth topic of Install Guide for recommendation on database growth.

- Database auto growth

  For systems expected to have large ION_Data databases (> 100 GB), change the default auto-growth setting to a fixed size rather than by percentage (Default is 10%). In this case, the database auto-grows when the database exceeds the preset container size. This ensures better control of database growth. Auto-growth events are expensive operations that slow down the performance of your database. Take the following preventive steps to ensure your auto-growth is in control:

  – Minimize future auto-growth events by monitoring the growth of your databases, and re-establishing auto-growth settings when a database growth profile changes.

  – Monitor auto-growth events so you can be alerted when your databases grow.

  – Consider defragmenting your database file system.

  See SQL Server Database Growth and Autogrowth Settings for more information.

Database maintenance

WinPM.Net uses databases to store information such as system configuration, data logs, and system event log messages. These databases must be maintained to preserve performance, manage disk space usage, and guard against data loss in case of database failure. Maintenance is the key to a healthy system that supports system longevity and future scalability. You must not ignore maintenance. Non-maintenance may lead to system downtime and you might need to rebuild the system from scratch.

NOTICE

LOSS OF DATA

- Back up the database at regular intervals.
- Back up the database before upgrading or migrating the system.
- Back up the database before trimming it.
- Back up the database before making manual database edits.
- Verify correct database behavior after making database or system changes.

Failure to follow these instructions can result in permanent loss of data.

Database maintenance for WinPM.Net includes the following key activities:

- Performance maintenance

- Accurate and up-to-date query statistics

- [Minimize index fragmentation](#)

- [Data archive and trim](#)

- [Database integrity check](#)

Based on the key activities, see [Consolidated recommendation for database maintenance](#).

Performance maintenance

For performance maintenance, enable and schedule the following daily maintenance tasks for all WinPM.Net systems of any configuration:

Database	Type of Data	Maintenance Tasks*		
		Maintenance (update statistics and index defragmentation)	Trim**	Size Notification
ApplicationModules	Web Applications related configuration data and system event log entries.	✓	✓	–
ION_Data	Historical power system data such as interval data logs, waveforms and alarms.	✓	✓	✓ **NOTE**: For systems with SQL Server Express, enable SQL Express Database Size Notification.
ION_Network	Device network and other system configuration data.	✓	–	–
ION_SystemLog	Non-Web Applications related system event log entries.	✓	✓	–

* See [Default maintenance task settings](#) for basic task definitions.
** Keep the last 30 days of data.

In Standalone WinPM.Net systems, the database maintenance tasks are pre-configured and scheduled to run automatically by default. For Distributed Database WinPM.Net systems, you need to configure the tasks and set up the schedules manually.

For more information, see the [Database maintenance](#) section in the Configuring chapter of this guide.

Accurate and up-to-date query statistics

SQL Server uses statistics to create query plans that improve query performance. As the database increases and holds more data, the statistics becomes less relevant over the time. Updating statistics ensures that queries run with relevant statistics.

For all systems, it is recommend to update database statistics daily. The following table shows the default database maintenance task schedules for standalone systems:

Database	Task*	Trigger Time
ApplicationModules	Maintenance	Daily at 03:30
ION_Data	Maintenance	Daily at 02:00
ION_Network	Maintenance	Daily at 07:30
ION_SystemLog	Maintenance	Daily at 07:05

* See Default maintenance task settings for basic task definitions.

In distributed systems, the database maintenance tasks are not pre-configured. You need to set up these tasks manually.

These scheduled tasks trigger the **DatabaseMaintenance.ps1** Windows PowerShell script. This script executes the local [Maintenance].[UpdateStatisticsAll] stored procedure in each database.

Check the WinPM.Net system logs and SQL Server logs to confirm that the scheduled tasks are completed successfully. The log might report errors if an issue arises. As the database grows, these jobs will take longer time to complete.

See Microsoft's SQL Server documentation on SQL Server Query Statistics for more information about query statistics.

For more information, see the Database maintenance section in the Configuring chapter of this guide.

Minimize index fragmentation

Database index maintenance is important to ensure optimal performance. When data is written to the databases, fragmentation occurs. Heavy fragmented indexes can degrade query performance and reduce WinPM.Net's response time.

To minimize index fragmentation, monitor the fragmentation regularly and perform re-indexing.

For small to medium size systems:

It is recommend to re-index daily. The following table shows the default database maintenance task schedules for standalone systems:

Database	Task*	Trigger Time
ApplicationModules	Maintenance	Daily at 03:30
ION_Data	Maintenance	Daily at 02:00
ION_Network	Maintenance	Daily at 07:30
ION_SystemLog	Maintenance	Daily at 07:05

* See Default maintenance task settings for basic task definitions.

In distributed systems, the database maintenance tasks are not pre-configured. You need to set up these tasks manually.

These scheduled tasks trigger the **DatabaseMaintenance.ps1** Windows PowerShell script. This script executes the local [Maintenance].[DefragIndexAll] stored procedure in each database.

Check the WinPM.Net system logs and SQL Server logs to confirm that the scheduled tasks are completed successfully. The log might report errors if an issue arises. As the database grows, these jobs will take longer to complete.

For large systems:

For large systems with ION_Data growing over 100 GB in size, it is important to review the frequency of the [ION_Data] database maintenance scheduled task and switch to running them manually.

Index fragmentation in ION_Data is unavoidable for large systems because of the amount of data written to the system and queried on a regular basis. Index fragmentation is also common when database reads exceeds database writes, that is, when WinPM.Net is configured with added components to move data from WinPM.Net to another non-WinPM.Net system.

For large systems, you need to monitor the index fragmentation daily. To monitor, run the following SQL commands against the database:

- DBCC SHOWCONTIG
- sys.dm_db_index_physical_stats

Both the commands generate a report on index fragmentation. The time it takes to complete these statements depend on the amount of data in the table and the level of fragmentation. The more fragmented the index, the longer the query will run. You should expect the query to return results within 1 to 20 minutes.

DBCC SHOWCONTIG

DBCC SHOWCONTIG displays fragmentation information for the data and indexes for specified tables.

> **NOTE**: This command applies to SQL Server 2008 to 2019 and is expected to be deprecated in a future version of SQL Server.

For ION_Data, run the following commands:

```
USE ION_Data

GO

DBCC SHOWCONTIG WITH TABLERESULTS, ALL_INDEXES
```

Review the output from DBCC SHOWCONTIG for the following three statistics:

Average Page Density:

Shows the accurate indication of how full your pages are. A high percentage means the pages are almost full, and a low percentage indicates much free space. This value should be compared to the fill factor setting specified when the index was created to decide whether or not the index is internally fragmented. The fill factor is the percentage of space on each leaf-level page that

should be filled with data, and it is applied only when the index is created, rebuilt or reorganized. If the Average Page Density and Fill Factor are close in value, then it would suggest that there is little index fragmentation.

Scan Density:

Shows the ratio between the Best Count of extents that should be necessary to read when scanning all the pages of the index, and the Actual Count of extents that was read. This percentage should be as close to 100% as possible. Defining an acceptable level is difficult, but anything under 75% definitely indicates external fragmentation.

Logical Scan Fragmentation:

Shows the ratio of pages that are out of logical order. The value should be as close to 0% as possible and anything over 10% indicates external fragmentation.

See [DBCC SHOWCONTIG (Transact-SQL)](#) for more information on DBCC SHOWCONTIG command.

sys.dm_db_index_physical_stats

sys.dm_db_index_physical_stats also returns size and fragmentation information for the data and indexes of the specified table or view in SQL Server. This command is available in SQL Server 2005 or later.

For ION_Data, run the following commands to show fragmentation details for tables with more than 100,000 rows and a fragmentation level of greater than 50%. Comment out the WHERE clause to show results for all table indices. The output is sorted by fragmentation level from highest to lowest.

```
USE ION_Data

GO

SELECT

      DB_NAME(db_id()) AS DatabaseName,

      OBJECT_NAME(object_id) AS TableName,

      object_id,

      index_id,

      index_type_desc,

      avg_fragmentation_in_percent,

      fragment_count,

      page_count,

      avg_page_space_used_in_percent,

      record_count

FROM

      sys.dm_db_index_physical_stats(db_id(),DEFAULT, DEFAULT,
DEFAULT, 'SAMPLED')
```

```
WHERE

      (record_count > 100000) AND (avg_fragmentation_in_percent > 50)

ORDER BY

      avg_fragmentation_in_percent DESC;
```

When reviewing the output from sys.dm_db_index_physical_stats, review the values in the avg_fragmentation_in_percent column. You should defragement the indexes, if the fragementation is 10% and above.

See sys.dm_db_index_physical_stats (Transact-SQL) for more information on sys.dm_db_index_physical_stats command.

Correcting index fragmentation

Any index with over 10% fragmentation should be corrected.

There are different corrective statements depending on the level of fragmentation. For WinPM.Net, select the statements as follows:

Fragmentation Percentage	Corrective Statement	Remarks
10 to 30%	ALTER INDEX REORGANIZE	Reorganizing an index uses minimal system resources and is an online operation, which means WinPM.Net can remain online during this operation.
> 30%	ALTER INDEX REBUILD WITH (ONLINE = OFF)	Rebuilding an index drops and re-creates the index. Depending on the type of index and Database Engine version, a rebuild operation can be done online or offline. For large indexes, it is recommended to perform this operation offline.

See Resolve index fragmentation by reorganizing or rebuilding indexes for more information on correcting index fragmentation.

Data archive and trim

The archive strategy supports data retention and compliance, while the trim strategy supports disaster recovery goals.

Archive and Trim shortens the backup process by keeping only business critical data in the live database and also reduces the resource demands in the disaster recovery efforts by shrinking the database to backup and restore.

Archiving is not recommended since it fractures the data into multiple databases. WinPM.Net is unable to query multiple databases at the same time to make comparisons in the data. It is possible to run reports against an archived database, but it can only be done on one database at a time.

However, the ION_Data database may need to be reduced in size for two reasons:

- SQL Server Express is used as the database engine, which has a limit of 10 GB for the .mdf file.

- SQL Server (Standard or Enterprise edition) is used as the database engine and the ION_ Data database has become so large that query performance (in Vista for example) is not acceptable to the WinPM.Net system users. It is also important to ensure that the ION_Data database is trimmed well within the hard drive size, as it can affect the operation of WinPM.Net.

For WinPM.Net systems with considerable database growth (medium to very large systems), it is important to consider frequent removal of older and lesser used data from ION_Data. WinPM.Net includes an ION_Data data archive maintenance task by default. The database archive task is pre-configured and disabled for standalone systems, while it must be manually added for distributed systems.

To determine if data archiving is needed, you must:

- [Understand the importance of archive and trim](#)

- [Determine the data retention needs](#)

- [Develop the archive and trim strategy](#)

Understand the importance of archive and trim

The purpose of the archive is to remove data from the live ION_Data database to reduce its overall size. An archive is a copy of a subset of data from the live ION_Data database based on a date range and the type of data (Data Records, Waveforms and Events).

When an ION_Data archive is created, it is attached to the SQL Server database engine so that its data is still accessible to Vista and Diagrams. However, the data is not available to other applications in the Web Applications component.

> **NOTE**: Data archival in WinPM.Net is different from the normal terminology of archiving. WinPM.Net's archive task does not remove data from the database, it only copies data to the archive. Once the data is archived in WinPM.Net, it cannot be re-imported back.

We recommend to manually trim the database of historical data after each archive task.

Determine the data retention needs

The live system only needs to hold as much data as needed for business. Consider the following questions along with the business use cases to determine the data retention needs:

- What is the oldest date of data that is needed for trends and alarms?

- Is it sufficient to review older data trends only through web reports or diagrams?

- What date range of data is needed for historical reporting – start and end dates?

- How often should data be archived?

Develop the archive and trim strategy

These questions determine key aspects of the archive and trim strategy, in particular:

- When to archive data from the live database?

- When to trim data from the live database?

- When to delete historical archives?

Example: Archive and Trim Strategy

The following diagram illustrates an example ION_Data archive and trim strategy for a small to medium system that started data collection in Q4 2016 and the strategy planned till 2021.
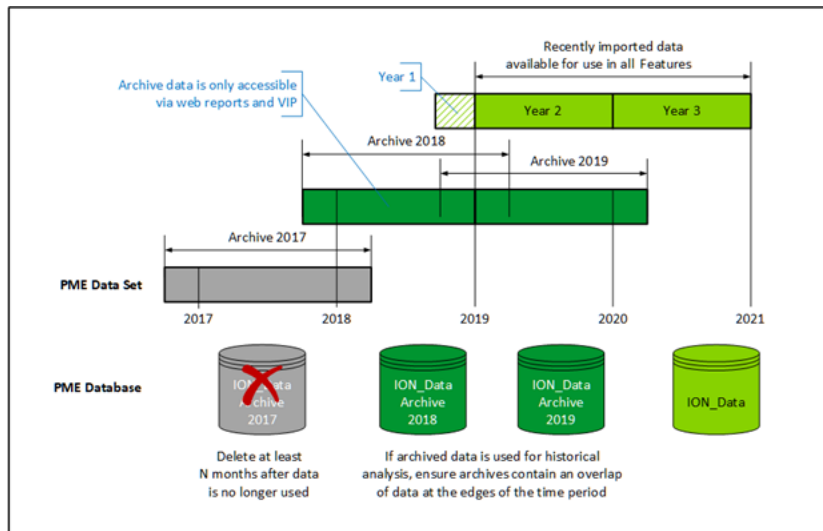
Based on the business needs, the data retention requirements are:

- Perform analysis with at least two calendar years of data in the main WinPM.Net system

- Reporting on the last 3 calendar years of data

- Data older than 3 calendar years can be deleted

The example strategy is as follows:

- Keep two calendar years of data in the main ION_Data database.

- Start archive activity at the end of Q1 of the third year.

> **NOTE**: There should be 2 years and 3 months of data in ION_Data.

In this example, archive data from start of Q4 Year 1 to end of Q1 Year 3 into the database called ION_Data Archive Year 1. This results in a new archive database named ION_Data_Archive_2017 containing data from 01 September 2016 to 01 April 2018.

- Backup the newly created archive database.

- Schedule the archive activity once per year starting at the end of Q1 of the current year.

> **NOTE**: Archive the older year of data with an additional 3 months on each side of the year of interest.

- After each successful archive, backup the newly created archive database.

- Schedule the trim activity to trim data in the last 3 months of the dataset from the main ION_Data database.

- Schedule the trim to run every 3 months.

- When an archive database contains data older than 3 years, mark the archive database for deletion.

- Delete the marked archive database after 12 months of non-use.

NOTE: This strategy creates an overlap of data for every year.

Recommendation for archive and trim

- Archive and trim the ION_Data database often and in small batches. Together these two tasks reduce the size of ION_Data and its backup files.

- Archive historical data often even when disk space is not an issue or when not using SQL express.

- Trim historical data in small batches and often.

- Always perform a database trim after verifying a new archive.

Recommended consolidated archive and trim plan for systems

The recommended consolidated archive plan of ION_Data database for the different purpose of WinPM.Net system are as follows:

Archive strategy parameter	Purpose of WinPM.Net system		
	Analysis & decision making (Capacity management, Energy usage analysis)	Real-time monitoring & troubleshooting (Electric distribution monitoring & alarming)	Critical large advanced distributed system
Maximum data retention (ION_Data and Archive databases)	4 years	1 year	4 years
Live data in ION_Data	3 years	1 year	2 years
ION_Data archives	Start after end of year 3	Not applicable	Start after end of year 2
Archive frequency	Annually	Not applicable	Every 3 months
Number of archives to store on server	1	Not applicable	8
Additional IT resources needed for archive databases*	Use database growth calculator to estimate size archive database	Not applicable	Use database growth calculator to estimate size archive database

*See Database growth topic of Install Guide for recommendation on database growth.

The recommended consolidated trim plan of ION_Data database for the different purpose of WinPM.Net system are as follows:

Trim strategy parameter	Purpose of WinPM.Net system		
	Analysis & decision making (Capacity management, Energy usage analysis)	Real-time monitoring & troubleshooting (Electric distribution monitoring & alarming)	Critical large advanced distributed system
Trim	Data with timestamps older than 3 years from today	Data with timestamps older than 1 year from today	Data with timestamps older than 2 years from today
Trim frequency	Monthly	Monthly	Monthly

Database integrity check

Database corruption is a rare event that is usually caused by inoperative hardware on the server. A database integrity check reviews the allocation and structural integrity of all objects in each database to ensure it is not corrupt.

Run DBCC CHECKDB in SQL Server Management Studio on all WinPM.Net related databases once per month or quarter.

Check for errors reported in the output of DBCC CHECKDB. A database with integrity displays the following at the end of the output.

```
CHECKDB found 0 allocation errors and 0 consistency errors in database
'ION_Data'.

DBCC execution completed. If DBCC printed error messages, contact your
system administrator.
```

See DBCC CHECKDB (Transact-SQL) for more information on database integrity.

Consolidated recommendation for database maintenance

Based on the key activities discussed, the following is the consolidated recommendation for database maintenance as per the systems:

For all systems (Small, Medium, and Large):

- Schedule the ApplicationModules database trim task to run daily.

- Schedule the ION_SystemLog database trim task to run daily.

- If the number of connected devices have increased over time, review hardware and hard drive space requirements at least once per year to ensure server specifications meets growing demand.

- Review the frequency of the ION_Data database maintenance task as the system grows. Reduce the frequency from daily to weekly to monthly as the database grows and the maintenance tasks (Update statistics and re-indexing) take longer to complete.

- Never shrink the database container; it causes fragmentation.

- Archive and trim the ION_Data database regularly and in batches, such as 3 to 12 month data sets. Refer to Historical Data Archive and Trim Strategy for more information.

- Commission WinPM.Net's default ION_Data archive Windows scheduled task.

- Develop a process and schedule for:

  - Verifying the newly created archive database

  - Backing up the newly created archive database

  - Deleting the archive database when necessary

For large ION_Data databases (> 100 GB):

Large ION_Data databases require additional effort to maintain because all the maintenance task might not run completely.

- Disable the ION_Data database maintenance task

  - With large systems, index fragmentation occurs quickly and is unavoidable.

  - Defragmentation takes lot of time and the performance gained because of defragmentation is comparatively less.

  - If you plan to rebuild indexes, ensure that you have an equivalent amount of free space as the database size.

- If ION_Data database increases over 100 GB unexpectedly, it can be due to following reasons:

  - Database fragmentation can occur when there are more database read than database write actions.

  - More data points are logged than usual and frequent logging is performed.

- Defragmenting indexes may require a lot of free hard drive space to allow reindexing to succeed. Review hard drive space requirements.

System health review

You can adjust the system for optimal performance by monitoring the state of a system.

Frequent review of system health is to ensure optimal system health over the long term. This task involves identifying and resolving potential issues. It is recommended to perform system health checks at least once per month or quarter depending on the amount of data flowing into the system and how often the system is used.

It is recommended to use the following approach to set up regular system health checks:

1. Determine which system health checks are appropriate. The following is a list of system health checks appropriate for most of the WinPM.Net system. Customized WinPM.Net systems might include additional checks or have some removed, however all system health checks listed should be considered:

List of checks	Application Server	Database Server
Anti-malware	✓	✓

List of checks	Application Server	Database Server
Services running	✓	-
Message queues	✓	-
Device communication	✓	-
Processor usage	✓	✓
Memory usage	✓	✓
Disk usage	✓	✓
File system growth	✓	✓
File system fragmentation	✓	✓
Log files	✓	✓
Windows scheduled task history and status	✓	✓
Database growth	-	✓
Database fragmentation	-	✓
Database integrity	-	✓
Database backup	-	✓
Software licensing	✓	✓
Software updates	✓	✓

2. Identify and document how the above information can be collected for the system health review. Note the following tools:

   – WinPM.Net Diagnostic Tool - Install and deploy this tool to obtain a snapshot of the current state of the system.

   – WinPM.Net Diagnostics Viewer. See Diagnostics Viewer for more information.

3. Create a template system health report. This report should contain at least the following information:

   – Report date

   – Contact information

   – A list of each system health check with the following information for each line item:

      – Status – Passed, Caution, or Failed

      – Description of contributing factor to the given status

      – Recommended action

4. Determine a storage location for system health reports. Reports should be stored in a consistent location and they should be accessible by administrators and support users.

5. Create an initial system health report.

6. In the location created in step 4, save the template report and the initial system health report.

The following table provides the information on list of system checks on why the check is required and what you need to check:

List of checks	Why	What to check
Anti-Malware	If the WinPM.Net server has an internet connection, it is at risk for viruses and malware. The anti-malware software should be monitoring for threats in real-time and running full scans once per month.	• Check for warnings and threats. • If threats were found, were the infected files quarantined? • Check that the latest anti-malware definitions are installed.
Services running	These services are core of WinPM.Net and must always be running.	• Use Windows Services, Windows Event Logs and Diagnostics Viewer – Service Diagnostics to ensure all ION, SQL Server and IIS services are running. • If they are stopped, investigate logs for root cause. See Diagnostics Viewer for more information.
Message queues	• Log Inserter writes log data into a message queue instead of writing it to SQL Server directly. Another process (the Log Subsystem Router Service) reads the messages from the queue and writes the data to SQL Server. • The message queues should be at or near zero the majority of the time. One indicator of poor system health is when WinPM.Net is operating in steady-state and the message queue size stays above zero for any queue for an extended period of time.	Use Diagnostics Viewer – Log Pipeline Service to check status of WinPM.Net MSMQs. See Diagnostics Viewer for more information.

List of checks	Why	What to check
Device communication	Networking issues can lead to communication loss with devices. For devices without onboard logging, communication loss means data loss. Monitor device communications often to ensure the expected devices are communicating.	• Use Diagnostics Viewer – Communication Diagnostics to check for communication issues, such as Timeouts or Log Inserter issues. • Repeated and frequent timeouts are common for long daisy chains – this is a sign that the loop performance needs to be assessed. • The LogInserter service diagnostic will reveal which devices cannot log data at the device level, and which DataRecorders have issues (reference LogHandle column). See Diagnostics Viewer for more information.
Processor usage	CPU usage over time should be less than 80%.	• Is CPU trend showing at least 20% free? • Are processes using and releasing CPU resources? In Windows Resource Monitor, track the following object counters: • Processor: % Privileged Time • Processor: %User Time • System: Processor Queue Length See Monitor CPU Usage for more information.
Memory usage	Prevent low memory problems by applying the appropriate server resources (RAM, CPU) as the system grows. Take into consideration future extensions of the system and upgrades, which could be performed in-place if the server is prepared ahead. Monitor memory usage to confirm that it is within range.	Use Windows Resource Monitor to track the following object counters over time to determine normal usage and identify issues: • Memory: Available Bytes • Memory: Pages/sec See Monitor Memory Usage for more information.

List of checks	Why	What to check
Disk usage	There is a risk of disk I/O issues particularly in WinPM.Net systems with large ION_Data databases.	Use Windows Resource Monitor to track the following object counters for each disk: Primary • PhysicalDisk: Avg. Disk sec/Write • PhysicalDisk: Avg. Disk sec/Read Secondary • PhysicalDisk: Avg. Disk Queue Length • PhysicalDisk: Disk Bytes/sec • PhysicalDisk: Disk Transfers/sec Track these counters over time to determine normal usage and identify issues. See Monitoring Disk Usage for more information.
File system growth	If a disk completely fills up, data loss occurs. You must ensure the all disks have adequate disk space for all maintenance tasks (defragmentation, backups, database reindexing). There should be at least 20 - 30% free disk space at all times for optimum performance. Possible causes of file system growth include database file growth, log file growth, data archives, and space used by 3rd party software. The best preventive measure is to track of the disk space usage and assess the growth over time. If the used space for a disk has consistently increased for several months and the percent free disk space is below 30% then action is required. Investigate the root cause of the growth, and develop a plan to either prevent more disk space from being used or for more disk space to be added.	• Ensure file system fragmentation job is running regularly • Check for unsustainable file system growth. If found, take preventive action to reduce the risk of a full disk, that is, add more disk space or adjust system configuration to reduce disk space usage.

List of checks	Why	What to check
File system fragmentation	If file system (or disk) fragmentation is greater than 40% on the database drive, SQL server experiences a thrashing/page faulting condition when manipulating a large volume of rows. For WinPM.Net, file system fragmentation usually results in database auto-growth. It is most common in small and medium databases because of the SQL express database size limit.	• Pre-allocate hard drive space for the ION_Data database. See Diagnostics Viewer for more information. • Check file system fragmentation on all drives used by WinPM.Net at least once per month • Schedule time and perform file system defragmentation if necessary. For standalone systems, ensure SQL Server services are stopped before defragmentation.
Log files	There are many logs that contain critical error information, non-critical error information, warnings, and informational messages. Logs can be a good source of information for how well a system is performing as well as gathering data for troubleshooting specific issues. IIS logs should be trimmed regularly.	Check: • ION_SystemLog • IIS Logs • Windows Event Logs • SQL Server Logs • Check for unexpected errors in the log files • Check for errors related to WinPM.Net components • Check log size to ensure total size is not excessively large (> 1 GB) • Archive historical logs if folder has too many log files. Sometimes certain irrelevant errors or warnings can be ignored. Any anomalous messages in these logs should be recorded and investigated.
Windows scheduled task history and status	WinPM.Net's default database maintenance tasks are configured as Windows Scheduled tasks. Ensure scheduled tasks are successfully launched at the scheduled time and completing.	• Check Windows Scheduled Task Logs, if any. • Check WinPM.Net System Logs for errors related to these tasks.

List of checks	Why	What to check
Database growth	Unexpected database growth can lead to poor performance. Usually significant database growth is a trigger. Unexpected database growth suggests possible excessive data logging due to device misconfiguration. For example, unexpected high-frequency logging (1 second logging intervals) or waveform logging when waveform data is not necessary.	Check database growth since the last system health check. Does the growth align with expectations?
Database fragmentation	Database fragmentation, if not addressed, is a common cause of poor system performance.	Check for index fragmentation over 10%
Database integrity	Database corruption is a rare event that is usually caused by inoperative hardware on the server. A database integrity check reviews the allocation and structural integrity of all objects in each database to ensure it is not corrupt.	• Run DBCC CHECKDB on all WinPM.Net related databases. • Check for errors reported in the output of DBCC CHECKDB.
Database backup	Confirm that the backup scheduled tasks are completing successfully.	• Review SQL Server Logs for errors related to each job. • Confirm that the expected database backup files exist, and that copies of the backups have been made to another media and off-site.
Software licensing	To ensure all WinPM.Net and SQL Server features are functional.	Check that the license is still valid.
Software updates	The latest updates ensure the system has the latest cybersecurity protection, and known software bugs are fixed.	Check to see if software is out-dated and identify when is the correct time for an upgrade.

Disaster recovery strategy

Disaster recovery requires planning and assessment to develop a strategy that meets both the business requirement and WinPM.Net system configuration. The disaster recovery strategy is the result of two objectives:

- Data retention - The amount of data required in the active system.

- System recovery - The minimal state of the system that should be recovered after a disaster and the acceptable limit of data and time loss.

Disasters can occur at any time and, if unprepared, such events can lead to data loss and service disruptions. Factors leading to system disasters include:

- Inoperative hardware

- Sudden power interruption / outage

- External threats such as malware, virus attacks, hacking

- Human errors such as accidental data deletion

- Implementation or upgrade issues

- Database corruption, such as a database exceeding the maximum expected size or allowable hard drive space

- Natural disasters (For example, earthquakes, fire, flood, storms, and so on)

Some SQL Server disasters cannot be prevented, so it is important to prepare a complete disaster recovery plan (DRP) to ensure minimal impact on service and data availability.

An effective disaster recovery plan should comprise:

- Identify disaster recovery objectives

- IT architecture and resources plan

- Backup plan

- Recovery plan

Developing the plan requires collaboration with the IT team, application champions (administrators, power users) and recovery experts.

NOTE: If you have limited time and resource to define the strategy, you can consider third-party products and services for assistance.

Identify disaster recovery objectives

The plan starts with identifying disaster recovery objectives. This includes the business aspect of the system. You might want to consider the value of the WinPM.Net system and the data it contains:

- What is the business cost of one day of downtime, both explicit and implied?

- What would be the result if an hour's, day's or month's worth of analysis, reporting, alarming and data were lost?

- What would be the result of a complete loss of the WinPM.Net system?

If your system is not critical, you may decide the best strategy is a simple one where a new WinPM.Net system is redeployed in the event of a disaster and device data is re-imported and you experience potentially irrelevant historical data loss. If your system is critical, you may develop a plan for a quick recovery with minimal data loss.

You must set a written expectation of what constitutes an acceptable loss. Consider the following questions:

- What is an acceptable level of data loss in your WinPM.Net systems?

  The answer to this question determines the **Recovery Point Objective (RPO)** objective. It is the maximum amount of recent data the business can lose when a disaster strikes. It helps to measure how much time can occur between your last data backup and the disaster without causing serious damage to your business. RPO is used to determine how often to perform data backups.

  For example, your backup schedule is set to daily at midnight and a disaster occurs at 8 AM. At the point of the disaster, you would have lost 8 hours' worth of data. If your RPO is one day of data then the loss of the last 8 hours of data is not an issue. However, if your RPO is one hour of data, then you must revise your backup schedule to at least one backup per hour.

- What is an acceptable recovery time?

  The answer to this question determines the **Recovery Time Objective (RTO)** objective. It is the amount of time the business can survive without the system after a disaster and before operations are restored to normal. It determines how quickly you need to recover the WinPM.Net system after a disaster.

  For example, if your RTO is 24 hours, you can wait up to 24 hours before the system must be available to users. If data and infrastructure are not recovered within 24 hours, the business might be impacted.

- What level of disaster should we be prepared for?

  Identify the possible disasters that could affect your WinPM.Net system and the level of impact of each disaster. For example: If your WinPM.Net system is an on-premise solution, you should prepare for disasters such as power loss, fire, flood, etc. If your WinPM.Net system is hosted on off-site servers in a data center, prepare for natural disasters but with low priority compared to cybersecurity risks.

IT architecture and resources plan

It is important to design the IT architecture so you can allocate the necessary hardware and networking resources in support of optimal performance and disaster recovery. Maintenance and backup activities often require additional resources (CPU, RAM and hard drive space) to perform and complete the activity. You can also prevent disasters using additional hardware. The recommended best practices are as follows:

- Hard drive space allocation

  Ensure there is enough hard drive space to perform backup operations and take at least two backup files. The spare hard drives can ensure minimal rebuild time. RAID arrays (commonly used on all WinPM.Net systems) can protect against disk damages. See Storage Performance and Availability topic of Install Guide for recommended hard drive sizing.

- Backup power

  UPS systems and redundant power supplies to servers can prevent server power
  interruption.

- Connection redundancy

  If available, redundant data links can protect critical data transmission when the
  communication cannot be established in the network.

- Standby servers

  With supporting infrastructure and cost, standby servers can provide another set of hardware
  that can replace the WinPM.Net system hardware in the event of an inoperative server. This
  approach is valuable when WinPM.Net is a critical system.

Backup plan

Creating backups are a key part of every WinPM.Net deployment. A backup solution unique to the
WinPM.Net deployment can be created based on the recovery objectives, the WinPM.Net
system, and available IT resources. The backup plan should comprise:

- Components backup

- Backup frequency

- Storage and retention of backup

- Test the backup

Once you have a strategy with details, document the details and supporting processes. Whenever
a system or process change occurs, review and update this document. Store the document
outside of the WinPM.Net server.

Components backup

The following table contains the components of a standard WinPM.Net system that must be
considered for backup:

Component	Name	Description
WinPM.Net Database	ION_Network	Sometimes called the NOM (Network Object Model), the ION_Network database stores device information, such as, device name, device type and connection address (for example, IP address and TCP/IP port or device/Modbus ID). It also contains information about the optional Application Module settings, other ION Servers, Sites, Dial Out Modems, and Connection Schedules. There is only one ION_Network per system

Component	Name	Description
WinPM.Net Database	Application Module	The Application_Modules database contains configuration settings (for example, layouts, colors, application events, and so on) and cached historical data for some of the Web Applications (for example, Dashboards and Trends).
WinPM.Net Database	ION_Data	The ION_Data database contains the historical data, events and waveforms from devices connected to the system. This includes: onboard logging configured on devices; and, PC-based logging configured in the device translators and the Virtual Processors.
WinPM.Net Database	ION_Data archive	The ION_Data archive databases contain historical data that have been sectioned off from the main ION_Data database.
System Database	master	The master database is the core system database for a SQL Server installation. It contains information such as SQL Server credentials and system configuration settings.
System Database	model	The model database is used as a template for all databases created on the SQL Server instance.
System Database	msdb	The msdb database is used by SQL Server Agent for scheduling alerts and jobs. msdb also contains history tables such as the backup and restore history tables.
WinPM.Net Files	WinPM.Net	The application folder is where all the program and configuration files for WinPM.Net are stored. By default this is "%Program Files%\Siemens\WinPM.Net"
System Files	SQL Server	The SQL Server folder is where all the program and configuration files for SQL Server, SQL Server Reporting Services, and SQL Server Agent are stored. By default, this is "%Program Files%\Microsoft SQL Server".
System Files	Windows registry	Contains configuration information for the entire server

Component	Name	Description
System and WinPM.Net Files	Full Server Backup	It is advised to take an image of the entire WinPM.Net application and database servers (excluding the actual database files – MDF, NDF, and LDF files). This backs up other important configuration information such as service credentials, security policies, and IIS setup. It can be important when simplifying and reducing the time taken for a system recovery.

All WinPM.Net databases should be backed up frequently and a full server backup should be taken upon system configuration changes (for example, Vista diagrams, updating device drivers, registry settings, VIP framework changes, and so on). Use Configuration Manager for performing the backup. See Configuration Manager topic in Install Guide for more information.

Database recovery model

When backing up databases it is important to choose an appropriate recovery model. The recovery model is a database property that controls how transactions are logged, whether the transaction log requires (and allows) backing up, and what types of restore operations are available.

WinPM.Net databases use one of two recovery models:

- Simple recovery model

  Complete database backup is taken and a restore can only be done up to the point when the backup was taken.

- Full recovery model

  Provides backup options such as differential, incremental, and transaction log. The restore can be done using different options.

All WinPM.Net databases are configured with the simple recovery model by default. The ION_ Data database recovery model should be updated to reflect your backup plan.

The recovery model is determined by comparing the disaster identification time with the backup schedule. For example, as per the following diagram:
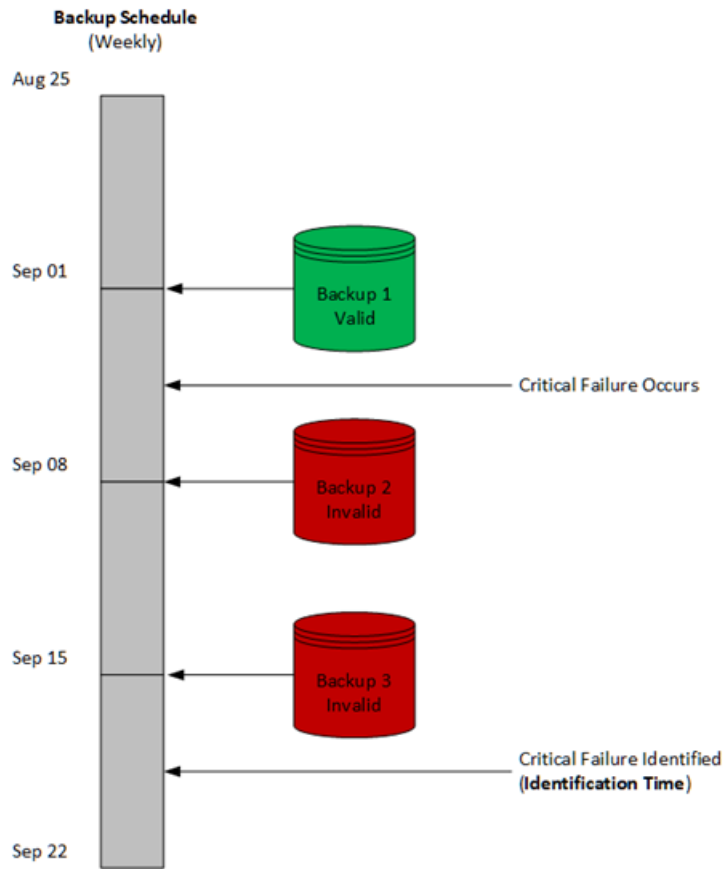
A system that is configured to have a single backup cannot be recovered. System is not accessed by users over the weekend, and becomes inoperative such that the automated backup jobs are still able to run. In this case, the backup would not be valid and there would be loss of the complete WinPM.Net system.

You can prevent this situation by setting the ION_Data database recovery model to Full, thus allowing more refined backup options.

In the case of critical WinPM.Net systems, consider:

* Being aware of your disaster identification time and adjust your backup schedule appropriately

* Using a full recovery model with several differential backups (advanced configuration)

* Keeping multiple backup copies on a rotational basis

The key benefit of the full recovery model is that it can restore a database exactly to any point in time since the last full backup was taken, including potentially to the point the disaster occurred, resulting in no data loss. It should only be used if simple recovery is not sufficient to meet the recovery needs as it incurs cost of performance and storage space.

Backup frequency

By default, WinPM.Net is configured to backup the ApplicationModules and ION_Network databases on a daily basis, while the ION_Data database is backed up once per week. This default configuration assumes that meters installed throughout the network have onboard memory and onboard logging enabled with a log of at least 14 days of data. This weekly

frequency balances the need for performance in steady state and disaster recovery preparation. Frequent transaction log backups can lead to an unnecessarily bloated LDF file, which can cause performance issues.

If your WinPM.Net system is critical, it is important to ensure you have a frequent backup strategy to support quick recovery. In this case, the recommended practices are:

- Set the ION_Data database recovery model to Full

- Schedule daily full backups

- Schedule hourly transaction log backups

- Continue to keep the last 2 full backup files on the server

- Increase hard drive storage space by the 2 x size of a ION_Data.MDF file for the additional transaction log backup files

- Keep the last 24 transaction log backup files on the server

The recommended backup configuration and frequency for WinPM.Net and system database are as follows:

Component	Name	Description	Recovery Model	Backup Frequency
WinPM.Net Database	ION_Network	All	Simple	Daily
WinPM.Net Database	Application Module	All	Simple	Daily
WinPM.Net Database	ION_Data	For systems with meters that have at least 14 days of onboard logging	Simple	Weekly
WinPM.Net Database *	ION_Data	For systems without onboard logging or for critical systems **NOTE**: Perform hourly transaction logs backup. Review data storage requirements for the additional transaction log backup files	Full	Daily
WinPM.Net Database	ION_Data archive	All	Simple	Upon creation
System Database	master	All	Simple	Daily
System Database	model	All	Full	As required
System Database	msdb	All	Simple	Daily

Additionally, all the components should be manually backed up after an update.

The recommended backup configuration and frequency for WinPM.Net and system files are as follows:

Component	Name	Backup frequency
WinPM.Net Files	WinPM.Net	• Backup upon significant system change • Use Configuration Manager to backup and archive WinPM.Net files. Be sure to deselect the database options. See Configuration Manager topic in Install Guide for more information.
System Files	SQL Server	• Backup upon significant system change • Backup "%PROGRAMFILES% Microsoft SQL Server" folder upon major system changes (hotfixes and upgrades)
System Files	Windows registry	Monthly and after a significant system change
System and WinPM.Net Files	Full Server Backup	• Annually or upon significant system change • Backup the entire WinPM.Net application and database servers (excluding MDF, NDF, and LDF database files) once a year and after each significant system change (upgrade)

Storage and retention of backup

In the WinPM.Net Planning stage, we recommend to have enough additional hard drive space to support at least three times the expected size of the main ION_Data (MDF) database. This estimation assumes that two backup files are stored on the production server.

We recommend the following storage and retention strategy:

- Follow the 3-2-1 Rule
    - Store backups locally on a RAID protected drive for the shortest amount of recovery time.
    - Store a copy of backups on a centralized set of disks so you can recover the backups on another server if the production SQL Server encounters a critical issue.
    - Store a copy of the backups off-site on external drives or in the cloud in case a site disaster occurs.
- Set up automated processes to backup and move files to separate locations.
- Maintain a reasonable set of backups off site and outside of the WinPM.Net servers. We recommend the following backup retention strategy. Check with your legal team on keeping certain amount of critical data in the event of a disaster.
    - 10 daily backups
    - 5 weekly backups
    - 6 monthly backups
    - 3 on-demand or annual backups
- Historical backup files should be stored off-site.

- Delete the old backup files on a regular basis in order to manage the storage cost.

Test the backup

A critical aspect to the backup strategy is to ensure that you can recover files from the backups. Prepare a test procedure to verify that the backup files contain the expected data and that the backups can actually be restored, that is, the backup files are not corrupt.

When practicing the restore procedure, ensure that you restore to a different server and at a different location.

This practice ensures that the recovery team:

- Knows the steps to follow when recovering from a data loss or disaster.

- Has existing infrastructure to support a recovery.

- Can stay calm and act efficiently in a real disaster situation.

Recovery plan

Backup files are worthless if they cannot be restored, so you must have a recovery plan with the goal of getting a recovered WinPM.Net system functional with minimum downtime and data loss. The disaster recovery objectives and backup and archive strategies help create a recovery strategy.

The most important point to remember when creating a recovery plan is that it is not valid until it is actually tested, and your recovery position is good as your last recovery test. Once you have a recovery plan, allocate some time to test your disaster recovery strategy. Be aware of who is executing the recovery as well. Do not assume that a specific person is available to restore the WinPM.Net system.

We recommend the following approach to developing a recovery strategy:

1. Set a time expectation for recovery (Recovery Time Objective).

2. Identify the necessary hardware, software, backup and archive files and types (full, differential, and log).

   Ensure resources – physical (servers, software, network) and personnel – are allocated and assigned, so they are readily available if a disaster strikes.

3. Document the entire recovery procedure.

   If you have a large recovery time window, such as 1 week, you may have enough time to contact WinPM.Net support team to assist in a recovery procedure. If you have a smaller time window then any WinPM.Net administrator (factoring in employee turnover) should be capable of performing a restore, so this procedure should be well documented. At a minimum, all backup and archive locations should be documented and accessible to any WinPM.Net administrator. Store the documentation outside of the WinPM.Net production servers.

   NOTE: Training WinPM.Net administrators and / or support staff on WinPM.Net disaster recovery may be important to ensure you have redundant personnel available.

4. Schedule system downtime and test the restore procedure. This is a necessary step to ensure the disaster recovery strategy is valid. Track the time the recovery procedure takes to verify your time expectation for recovery is valid. Take corrective action for any areas missed in your recovery documentation, backup, or archive strategies.

5. Progressively update recovery documentation after any major system changes are made that changes the restore procedure.

NOTE: See SQL Server Limitations on Restoring System Databases in cases where a full SQL server recovery is needed.

Recommended consolidated disaster recovery strategy plan

The recommended consolidated disaster recovery strategy plan of ION_Data database for the different purpose of WinPM.Net system are as follows:

Disaster recovery strategy parameter	Purpose of WinPM.Net system		
	Analysis & decision making (Capacity management, Energy usage analysis)	Real-time monitoring & troubleshooting (Electric distribution monitoring & alarming)	Critical large advanced distributed system
Device memory	On board logging (14 to 30 days)	None	None
Recovery point objective	Up to 1 week	Yesterday	As close to the point of disaster as possible
Recovery time objective	24 hours	36 hours	24 hours
Database recovery model	Simple	Simple	Full
Full backup with frequency	Full with weekly	Full with weekly	Full with daily
Additional backup with frequency	Not applicable	Differential with daily	Transaction logs with hourly
Number of full backup files to store on server	2	2	2
Number of additional backup files to store on server	Not applicable	2 to 6	Depends on how often the transaction log backup files are validated

Disaster recovery strategy parameter	Purpose of WinPM.Net system		
	Analysis & decision making (Capacity management, Energy usage analysis)	Real-time monitoring & troubleshooting (Electric distribution monitoring & alarming)	Critical large advanced distributed system
Additional IT resources needed for additional backup files*	Standard	Additional storage required for differential backup	Additional storage required for transaction log backup

*See Introduction chapter of Install Guide for recommended system sizing.

Chapter 12: Reference

This chapter contains specific information on technical details related to WinPM.Net. This information is referenced in the other sections of this Help.

You access the information in this chapter through links from other chapters.

Operating Environment

WinPM.Net supports the following environments and software:

NOTE: The operating system and SQL Server combination you choose must be supported by Microsoft. This applies to edition, version, and 32-/64-bit.

Software	Supported Versions
Operating system	Windows 10 Professional/Enterprise Windows Server 2012 Standard Windows Server 2012 R2 Standard/Enterprise Windows Server 2016 Standard Windows Server 2019 Standard
Database system**	SQL Server 2012 Express SQL Server 2014 Express SQL Server 2016 Express SQL Server 2017 Express SQL Server 2019 Express (included with WinPM.Net 10.0) SQL Server 2012 Standard/Enterprise/Business Intelligence SQL Server 2014 Standard/Enterprise/Business Intelligence SQL Server 2016 Standard/Enterprise/Business Intelligence SQL Server 2017 Standard/Enterprise/Business Intelligence SQL Server 2019 Standard/Enterprise/Business Intelligence
Virtual environment***	VMWare Workstation 10 VMWare ESX1 6.0 Oracle Virtual Box 5.0.4 Microsoft Hyper-V from Windows 8.1, Windows Server 2012 Citrix XenServer 6.2 Parallels Desktop 10 QEMU-KVM
Microsoft Excel	Microsoft Excel 2013, 2016, 365
Desktop Web browser	Google Chrome version 42 and later Mozilla Firefox version 35 and later Apple Safari versions 7 or 8 and later Microsoft Edge
Mobile Web browser	Safari on iOS8.3+ operating systems, Chrome on Android systems
.NET Framework	.NET 4.6 or higher

** WinPM.Net includes a free version of SQL Server Express. You have the option to install this Express version during the installation of WinPM.Net, if you don't want to use a different SQL Server.

*** You must configure virtual environments with a supported Windows operating system and SQL Server edition. It is possible to mix virtual and non-virtual environments for WinPM.Net server and clients.

WinPM.Net Windows services

All WinPM.Net applications without a user interface run as Windows services. The following table lists all WinPM.Net services:

Service Name	Startup Type	Log On Account	Description
WinPM.Net Application Modules Alarm Services Host	Manual	Local System *	Allows the Event Notification Module (ENM) to read alarms directly from the ION_Data database. Starts on demand from other services (for example, from the Event Notification Module).
WinPM.Net Application Modules Core Services Host	Automatic	Local System *	Hosts common web services used by the Web Applications component.
WinPM.Net Application Modules Data Services Host	Automatic	Local System *	Hosts web services that provide low-level access to system data (that is, real-time, historical, alarming, and authentication) for the Web Applications component.
WinPM.Net Application Modules Provider Engine Host	Automatic	Local System *	Hosts web services that provide data processing for the Web Applications component.
WinPM.Net Component Identifier Service	Manual	Local Service *	Locates local and remote product components. Starts shortly after startup by request of WinPM.Net Connection Management Service.
WinPM.Net Connection Management Service	Manual	Local Service *	Determines the connection status of sites and devices in the system, and handles allocation of resources such as modems. This service manages the state of site and device connectivity for the system. In order to establish the most appropriate state for the system, each connection and disconnection request is evaluated against the overall state of the system and availability of communications channels. Starts shortly after startup by request of WinPM.Net Network Router Service.

Service Name	Startup Type	Log On Account	Description
WinPM.Net Event Watcher Service	Automatic	Local Service *	Monitors system events for conditions specified in Event Watcher Manager.
WinPM.Net Log Inserter Service	Automatic (Delayed Start)	Local Service *	Provides historical data collection for the power monitoring system (that is, devices and Virtual Processor), and stores it in the ION_Data database.
WinPM.Net Log Subsystem Router Service	Automatic (Delayed Start)	Local Service *	Transfers data received from power monitoring devices to storage and processing.
WinPM.Net Network Router Service	Automatic	Local Service *	Routes all ION requests between the software components, such as client workstations, the Real Time Data Service, Log Inserter, and the Query Server. The service dynamically detects changes to the network configuration, including the addition of new servers. It can also recognize new software nodes, such as Vista, that are added to an existing server.
WinPM.Net OPC Data Access Server	Manual	Local Service *	Serves real-time OPC data (OPC DA) to OPC client applications. Starts on an OPC client request for data, if the OPC license has been activated.
WinPM.Net PQDIF Exporter Service	Manual	Local Service *	Translates power quality data from the ION_Data database into PQDIF file format and manages scheduled PQDIF exports.
WinPM.Net Query Service	Automatic	Local Service *	Provides historical data retrieval from the ION_Data database for client applications (for example, Vista and Diagrams).
WinPM.Net Real Time Data Service	Automatic	Local Service *	Manages and provides access to real-time data for all client applications (Vista, Diagrams, Trends, and so on).
WinPM.Net Report Subscription Service	Automatic (Delayed Start)	Network Service *	Runs Reports subscriptions according to user-defined schedules. Starts several minutes after the server starts.

Service Name	Startup Type	Log On Account	Description
WinPM.Net Site Service	Automatic	Local Service *	Manages communication links to and from the product. WinPM.Net Site Service is responsible for handling packet communications to system devices and controlling direct device communications. The service reacts to changes in network configuration: for example, changes to certain channels, configuration parameters, ports, or device parameters can often interrupt a connection.
WinPM.Net Software Data Processing Service	Automatic (Delayed Start)	Local Service *	Performs evaluations based on real time data from the power monitoring system.
WinPM.Net Virtual Processor Service - NVIP.DEFAULT	Automatic	Local Service *	Provides aggregation, control, and mathematical analysis of power monitoring system data.
WinPM.Net XML Subscription Service	Automatic	Local Service *	Manages subscriptions to XML data for Vista user diagrams. This service is used only by the Diagrams application. When you open a Vista user diagram in a web browser, the WinPM.Net XML Subscription Service creates a subscription and delivers the real-time data in XML format.
WinPM.Net XML Subscription Store Service	Automatic	Local Service *	Stores XML data subscriptions for the power monitoring devices on the network. This service is used only by the Diagrams application.
SQL Server (ION)	Automatic	Local System	Provides storage, processing and controlled access of data, and rapid transaction processing for the ION_Data, ION_Network, ION_SystemLog, and the ApplicationModules databases.

* When WinPM.Net is installed with Windows Integrated Authentication, the Windows account that is used to access the database is also used to run the WinPM.Net services.

WinPM.Net Databases

WinPM.Net uses four databases to store device communication parameters, system configuration settings, and logged historical data.

ION_Network database

Sometimes called the NOM (Network Object Model), the ION_Network database stores device information, such as, device name, device type and connection address (for example, IP address and TCP/IP port or device/Modbus ID). It also contains information about the optional Application Module settings, other ION Servers, Sites, Dial Out Modems, and Connection Schedules. There is only one ION_Network per system.

ION_Data database

The ION_Data database contains the historical data, events and waveforms from devices connected to the system. This includes: onboard logging configured on devices; and, PC-based logging configured in the device translators and the Virtual Processors.

Application Modules database

The Application_Modules database contains configuration settings (for example, layouts, colors, application events, and so on) and cached historical data for some of the Web Applications (for example, Dashboards and Trends).

ION_System log database

The ION_SystemLog database holds system events and their timestamps, which is accessible to view in Management Console. Event priorities can range from 0-255 and are grouped into Diagnostic (0 - 5), Information (6 - 20), Warning (21 - 63), Error (64 - 191), and Critical (192 - 255) categories. System events can include:

- ION Service stopped or is starting or user connection to an ION Service is lost.

- Device has been declared offline / online.

- ION Site Service connected, disconnected or failed to connect to a Site.

- ION User logs on / off Vista or Designer.

- ION User saves a Vista or Designer node diagram.

- Plus many other Warnings and Errors relating to WinPM.Net system functions.

Default device type support

The following tables list the pre-configured device types in WinPM.Net 10.0.

Device Type
4300
4700
4720
9410
9810
3300 ACM
3710 ACM

Device Type
3720 ACM
3800 RTU
7SJ62
7SJ64
7UM62
ACCESS 9200
ACCESS 9200 MODBUS
ACCESS 9300
ACCESS 9330
ACCESS 9340
ACCESS 9350
ACCESS 9360
ACCESS 9500
ACCESS 9500 RTU
ACCESS 9510
ACCESS 9510 RTU
ACCESS 9600
ACCESS 9610
ACCESS 9700
ASCO 7000 Controller
BCPM
CM3000 Series
CM4000 Series
Digi Trip 1150
ION 6200
ION 6200 MODBUS
ION 7300
ION 7330
ION 7350
ION 7500
ION 7500 RTU
ION 7550
ION 7550 RTU
ION 7600

Device Type
ION 7650
ION 7700
ION 8300
ION 8400
ION 8500
ION 8600
ION 8600C
ION 8650
ION 8650C
ION 8800
ION 8800C
ION 9000
IQ Analyzer
ISGS
PAC3100
PAC3200
PAC4200
PM200 Series
PM700 Series
PM800 Series
PM8000
PM810 LE
S7-I/O
SAMMS
SB-EC
SER 3200
Static Trip III
VL 555
VL 586
VL Breaker
WL Breaker

VIP Modbus Slave port settings

> **NOTE**: To view or change these settings in your system, open Management Console and go to **Tools** > **System** > **ION Virtual Processor Setup**. For the VIP for which the Modbus Slave is configured, expand **Modbus Network** > **Slave Ports**.

Serial Port settings:

Setting	Range
Unit ID	1 - 247 (default: 100)
Receive Timeout (ms)	100 - 10000 (default: 1800)
Transmit Delay (ms)	10 - 60000 (default: 50)
Baud Rate	300 - 115200 (default: 9600)
Parity	NONE, ODD, EVEN, MARK, SPACE (default: NONE)
Stop Bits	1, 1.5, 2 (default: 1)

Modbus TCP settings:

Setting	Range	Notes
Unit ID	1 - 247 (default: 100)	
Receive Timeout (ms)	100 - 10000 (default: 10000)	
Transmit Delay (ms)	10 - 60000 (default: 50)	
Listening Port	1 - 9999 (default: 502)	It is possible that other Modbus Slaves are configured on the system, such as the ION Modbus Gateway Service that is used for breaker aging calculations. Configure different listening ports for the different Modbus Slaves. Each Modbus Slave needs its own, unique listening port.
Max Number Connections Allowed	1 - 32 (default: 5)	Configure this setting to only allow as many connections as needed for your application. Allowing more connections than needed increases that attack surface for unauthorized system access. Keeping the attack surface as small as possible is a basic security measure.

Tunneling of OPC data

The networking protocol for OPC is DCOM. DCOM is difficult to configure, responds poorly to network interruptions, produces high network traffic, and has significant security weaknesses.

OPC tunneling applications are designed to overcome these limitations when using OPC over a network. The goal of OPC tunneling is to eliminate DCOM by replacing it with a network protocol like TCP. Instead of connecting the OPC client to a networked OPC server, the client program connects to a local OPC tunneling application, which acts as a local OPC server. The tunneling application accepts requests from the OPC client and converts them to TCP messages, which are then sent across the network to a companion tunneling application on the OPC server computer. There the request is converted back to OPC and is sent to the OPC server application for processing. Any response from the server is sent back across the tunnel to the OPC client application in the same manner.

Diagram: OPC data is sent over the network by OPC tunneler using common network protocols.

OPC tunneling products are available from several vendors, including Kepware and Matrikon.

When selecting OPC tunneling applications, look for products that

- keep the synchronous OPC transactions local to the client and server, with an asynchronous tunnel connection

- provide link monitoring

- handle network disruptions effectively

- meet your security needs

Database maintenance task definitions

The following are high level definitions of WinPM.Net relevant database maintenance tasks.

Archive

Database archiving copies older data from the operational database into a separate, new database. The goal of archiving is to keep data safe for future reference. Data is typically archived based on calendar time intervals, for example by month or by year.

The WinPM.Net archive task creates a new archive database each time the task is run. Each new archive database is attached to SQL server and is available to be accessed by WinPM.Net.

> **NOTE**: The WinPM.Net archive task does not trim data from the operational database; it only makes a copy of the archived data, leaving the original data in the operational database. See Considerations for trimming archived data from ION_Data for important information on this topic.

Backup

Backing up a database creates a copy of the operational database. The goal of a backup is to have an identical duplicate of the operational database that can be used to restore the system in case the operational database becomes nonfunctional. Database backups should be created on

a regular basis, for example daily or weekly.

Maintenance

The WinPM.Net database Maintenance task defragments the database and updates the database statistics. The goal of these activities is to maintain database performance. Maintenance tasks should be run on a regular basis, for example daily.

Size Notification

The size notification task is used to monitor the size of the database and to notify users when a certain size threshold is reached. When the size threshold is reached, the task logs a system log event message and triggers a Critical alarm in WinPM.Net every time the task runs.

> **NOTE**: The Size Notification task is only configured for systems using SQL Server Express, which has a maximum database size limitation of 10 GB.

Trim

Trimming a database deletes data from the database. The goal of trimming is to prevent the database from growing to a size that could affect system performance. Databases should be trimmed on a regular basis, for example daily or weekly. For WinPM.Net only the system log databases are trimmed.

Considerations for trimming archived data from ION_Data

When archiving and then trimming data from the ION_Data database, you are moving this data from the operational database into an archive store for long-term retention. This data is then no longer available in the ION_Data database for analysis in WinPM.Net. WinPM.Net has very limited access to archived data.

We recommend that you only trim archived data from the ION_Data operational database, when:

- It approaches its size limit, for example 10 GB for a SQL Server Express database.
- It reaches a size that impacts query performance.
- The database drive is low on available free space and you cannot switch to a larger drive.

When you trim data from an SQL database, the database file size remains unchanged. After the trim, the database will first fill the new free space before growing the database file size again. To reduce the database file size after trimming, Shrink the database, using standard SQL Server tools.

> **NOTE**: The WinPM.Net archive task does not trim the database; it only copies data to the archive.

Archive data access in WinPM.Net:

Application	Archive Data Access
Vista	Yes

Application	Archive Data Access
Reports	Can access either data from the operational database or from an archive database but not both at the same time.
Dashboards	No
Diagrams	Yes
Trends	No
Alarms	No

Default maintenance task settings

The default WinPM.Net database maintenance tasks are defined as Windows PowerShell scripts, and scheduled and executed using Task Scheduler in Windows. The following table shows the different configuration settings for these tasks, their defaults, and where they are configured:

Setting	Location	Default Value	Comments
Windows account used to run the task	Task Scheduler	User account that is creating the task.	Use IONMaintenance for Standalone systems. Create a new, dedicated account for Distributed systems.
Task trigger times	Task Scheduler	See default task schedules.	The default schedules are configured automatically for Standalone systems. The schedules for Distributed systems must be configured manually.
Task enabling or disabling	Task Scheduler	See default task schedules.	n/a
Backup folder location	PowerShell script : Configuration.ps1 Variable: $locationForBackupFiles	...\ WinPM.Net \Database\Backup*	Sets the folder to which the database backups are saved. The backup script will create the following subfolders in this location: ...\Data for ION_Data ...\Network for ION_Network ...\SystemLog for ION_SystemLog ...\Applications for ApplicationModules
Archive folder location	PowerShell script : Configuration.ps1 Variable: $locationForArchiveDBFiles	...\ WinPM.Net \Database\Archive*	Sets the folder to which the database archives are saved.
Data to keep when trimming	PowerShell script : Configuration.ps1 Variable: $diagnosticsDaysToKeep	30	Sets how many days' worth of data is left in the database after trimming.
Database size (max)**	PowerShell script : Configuration.ps1 Variable: $maximumDatabaseSizeIn Gigabytes	9	Sets the maximum database size. This value is used by the size notification task to assess what percentage of database space has been used. The maximum size for a SQL Express database is 10GB. The maximum size in the script is set to 9GB to allow for a 1GB warning buffer before the database stops logging data.
Database size notification limit**	PowerShell script : Configuration.ps1 Variable: $databaseSizeNotification ThresholdPercentage	85	Sets the threshold for when a database size notification will be issued.

* Default setting for Standalone systems.

** The database size notification is only configured for systems with SQL Server Express databases.

Using IONMaintenance for database maintenance tasks

IONMaintenance is a Windows user account that is created automatically when WinPM.Net is installed in a Standalone architecture. This account is used to run the pre-configured WinPM.Net database maintenance tasks in Task Scheduler in Windows. To make changes to the task schedules in Task Scheduler, you need to enter the password for the IONMaintenance account. By default, the password for the account is generated automatically by the installer and cannot be retrieved. To have access to the password, you need to set a custom password for this account.

To set a custom password for IONMaintenance, run the WinPM.Net installer in maintenance mode and select **Reset Accounts** to start the account password reset procedure. Follow the installer instructions to reset the password.

NOTE: If you set a custom password for the Windows Accounts during the installation of WinPM.Net, then this is the password used by IONMaintenance and you can use this password to make changes to the task schedules in Task Scheduler.

NOTE: IONMaintenance shares the same password with IONUser, another account that is generated automatically by the installer and which is used for report subscriptions.

NOTE: If you change the password for the Windows accounts, the password you are providing must comply with the password policies of the Windows server and the SQL server. The software installer cannot validate the password at the time you enter it. If the password is not valid, the password reset and reconfiguration will not be successful. In that case, complete the reconfiguration, skipping any unsuccessful steps.

Database maintenance account requirements

WinPM.Net uses Task Scheduler in Windows for the scheduling and execution of database maintenance tasks. Task Scheduler requires a Windows account to run the tasks. In Standalone WinPM.Net systems, an account, **IONMaintenance**, is created by the installer and automatically assigned to the Task Manager tasks. In Distributed WinPM.Net systems you need to create an account manually. This account must meet the following minimum requirements:

In Windows on the computer where the database server is installed, the account:

- must be a member of the Users group.
- must have the following Windows policy settings: **Log on as a batch**; **Deny log on locally**.

In SQL Server, the account:

- must have a **public** server role
- (for archive task only) must have a **sysadmin** server role
- must have the following role memberships for the WinPM.Net databases (ION_Data, ION_Network, ION_SystemLog, ApplicationModules):
  - db_backupoperator
  - db_ddladmin

– Maintenance

– public

> **NOTE**: You will need the password for this account during the initial task setup, and later if you want to edit the tasks in Task Manager in the future.

Changes in security settings of device

The security settings for the following device types have changed starting the firmware version 3.0.0:

- 9810 series devices
- 9410 series devices

The changes in the device security settings are:

- Standard security mode is removed
- Legacy security support is an interim solution for users that use standard security mode.

Because of these changes, configuring the devices and viewing the diagram with firmware version 3.0.0 or higher requires changes in the WinPM.Net.

To access an existing device in Management Console after firmware upgrade:

1. Right-click the device that has Advanced Security enabled and select **Security** to open the **Meter Security Settings** dialog.

2. From the Software Access drop down list, select **User1**.

3. Click **Match Password** and type the password in the Password field then re-type it in the **Confirm Password** field.

4. Select the check box if you want to allow the software to send secure time synchronization signals to the device. Clear the check box if you do not want to send secure time synchronization signals to the device.

5. Click **OK**.

To configure the security module in Designer for an existing device after firmware upgrade:

> **NOTE**: The **Change Standard Meter Security** option is disabled and hence standard security can not be enabled.

To enable advanced security,

1. Open a node diagram.

2. Right-click the **Security Options** module icon to open the module setup.

3. Highlight the **SOP1 Legacy Security Support** register and click **Modify**.

4. Select **Disabled** in the options drop down and click **OK** in the Modify Enumerator Register dialog.

5. Click **OK** in the ION Module Setup dialog to close it.

6. Send and save the changes in Designer.

To perform control object function of diagram in Vista for an existing device after firmware upgrade:

1. Enter the username and password. See Controlling system functions for more information.

Device Type Editor User Interface

Main user interface
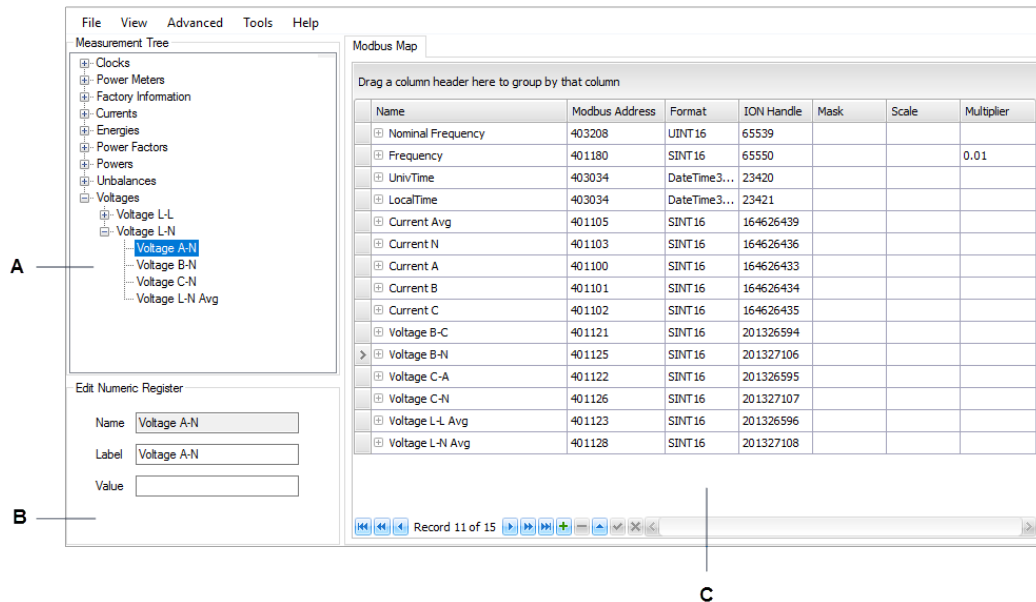
| A | Measurement Tree pane | B | String Register Details pane | C | Modbus/OPC Map pane |

Menu bar

Menu	Options
File	**New**: Create a new Modbus or OPC device type. **Open**: Open an existing device type. **Save**: Save the current device type. **Save As**: Save the current device type to a different location. **Export**: Export the current device type as text file in csv format **Exit**: Close the Device Type Editor.

View	**Measurement Tree**: Show or hide the Measurement Tree display pane **Register Detail**: Show or hide the String Register Details display pane **Register Map**: Show or hide the Modbus Register Map display pane. Show or hide the Enumerations, Address Blocks, Invalid Addresses, and Device Info tabs.
Advanced	Show or hide the Handle Exclusions, Transaction Configuration, Time Sync Info, and Plugin Info tabs.
Tools	**Configure Software Logging**: Open the Configure Logging and Calculation tool **Expose OPC Measurements**: Update the OPC measurement definition in the system **Clean Measurement Tree**: Remove any unused managers, modules, registers from the tree **Install Device Type**: Open the Device Type Installer
Help	**Device Type Editor Help**: Open the Device Type Editor help file.

Context Menus

Right-click on a manager, module, or register in the ION Tree to open a context menu. The context menu provides access to the actions you can perform on the Measurement Tree, such as adding and editing managers, modules, and registers.

Right-click anywhere inside the Modbus/OPC Map table to open a context menu. The context menu provides access to the actions you can perform on the Map , such as inserting or deleting rows.

Right-click anywhere inside the Map table header row to open a context menu. The context menu provides access to actions related to the table format, such as sorting of the table content or showing or hiding table columns.

Configuring Logging and Calculation User Interface

Use the **Configure Logging and Calculation** utility to:

- Enable register logging, including setting log frequency.

- Enable low (minimum), high (maximum) and mean (average) calculations and logging.

- Configure stale data settings.

- Append downstream device information to labels.

NOTE: Only numeric output registers can be calculated and logged. All other types of registers (for example, enumerated) do not appear on the Configure Logging and Calculation screen.

Screen overview
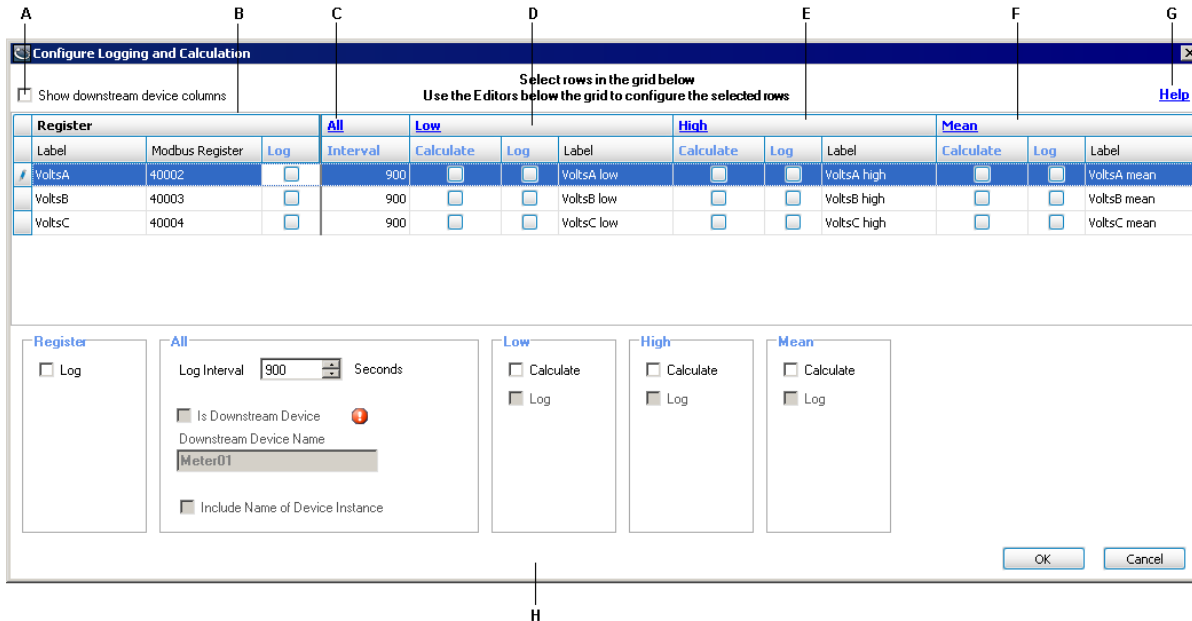
A	Select this check box to make the downstream device columns appear				
B	Register area	C	Stale data link	D	Low area
E	High area	F	Mean area	G	Help link
H	Global register edit area. Use this area to globally edit all rows selected in the upper pane.				

All the register information from the device type in the Device Type Editor is pre-populated in this table when you open the application.

Show downstream device columns: Select this to show all the columns for selecting and setting the downstream device properties for registers.

Register area: Use this area to individually configure registers.

Low area: Use this area to enable calculation and set logging of low values.

High area: Use this area to enable calculation and set logging of high values.

Mean area: Use this area to enable calculation and set logging of mean values.

Global register edit area: Use the bottom half of the screen to edit individual registers or multiple registers, after selecting them in the list.

Help link: Click this link to open the Device Type Editor online help.

Stale data link (All): Click this link to open the Configure Stale Data Settings dialog.

Configuring register logging

To edit registers on an **individual** basis, select the row of the register and edit the cells in the row or use the editing tools in the Global area.

To edit **more than one** register, select the rows you want to configure similarly and use the editing tools in the Global area.

Log: Select to enable logging of the register.

Interval: This value determines the logging frequency for the register. Note that this value also controls the reset interval for the min, max and averaging modules, even if these calculations are not being logged. The valid range is 1 to 4,000,000 seconds.

Configuring Low, High and Mean Calculation and Logging

To edit the suffix used for Low, High and Mean, click on the applicable link. For example, if you change the suffix to Maximum, VoltsA **High** becomes VoltsA **Maximum**.

Low

Select **Calculate** to enable calculation of low (minimum) values for this register. Select **Log** to enable logging of this value. Note how the label is appended to reflect that this is the low value.

High

Select **Calculate** to enable calculation of high (maximum) values for this register. Select **Log** to enable logging of this value. Note how the label is appended to reflect that this is the high value.

Mean

Select **Calculate** to enable calculation of mean (average) values for this register. Select **Log** to enable logging of this value. Note how the label is appended to reflect that this is the mean value.

Configuring Stale Data settings

The "staleness" of data is measured from time data was last read from a physical device until the current time.

Click the **All** (Stale Data) link to open the **Configure Stale Data Settings** dialog.

Mark data stale after: Enter the time you want to elapse before data from this register is marked stale in the data recorder. The valid range is 0 to 7200 seconds.

When data is stale: Select how you want stale data values logged. The setting "log an empty row" logs an empty row in the data recorder, while "use last known value" logs the last non-stale value in the data recorder until the end of the current interval.

> **NOTE**: Stale data settings are global (they apply to all registers) for a device type.

Configuring downstream device functionality

You must select the **Show downstream device columns** check box in the **Configure Logging and Calculation** window before you can edit downstream device functionality.

Show downstream device columns: Select this check box to make the downstream device columns appear.

Is Downstream Device: Select this checkbox to indicate the register belongs to a downstream device.

Downstream Device Name: Enter the name of the downstream device in this field. The register label is appended with this name. For example, if the device name is Meter01, the register label becomes VoltsA@Meter01.

NOTE: Using this option **without** also using the Include Name of Device Instance option causes multiple devices to appear as a single device in your energy management system. Do not use this option on its own unless only a single instance of this device is present in your system.

Include Name of Device Instance: Select this check box to append the actual device instance name to the register label. The placeholder !!DeviceName!! is then added to the label — the name of the actual device instance gets inserted when it is created.

For example, if the device instance is named Device01:

VoltsA@**!!DeviceName!!**Meter01

is replaced with

VoltsA@**Device01**Meter01

This allows for the creation of unique names within WinPM.Net, as well as enabling each device instance to appear in the product as multiple virtual devices.

For more information on downstream devices and naming conventions, see Downstream Device Assistant in the WinPM.Net User Guide.

Saving and exiting

To exit the Configure Logging and Calculation screen:

- Click **OK** to save your settings.
- Click **Cancel** to exit without saving.

You are returned to the main Device Type Editor console screen.

Modbus Data Formats

WinPM.Net supports the following data formats:

Format Type	Description
SINT16	A 16-bit signed value in 2-1 (or big ENDIAN) format. The high order byte is first, the low-order byte second.
UINT16	A 16-bit unsigned value in 2-1 (or big ENDIAN) format. The high order byte is first, the low-order byte second.
SINT32 or S32-4321	A 32-bit signed value returned in two consecutive 16-bit registers. The high-order word is returned in the first register, the low order word in the second. In effect, the 4 bytes are returned in 4-3-2-1 (or big ENDIAN) format.
UINT32 or U32-4321	A 32-bit unsigned value returned in two consecutive 16-bit registers. The high-order word is returned in the first register, the low-order word in the second. In effect, the 4 bytes are returned in 4-3-2-1 (or big ENDIAN) format.

Format Type	Description
S32-2143	A 32-bit signed value returned in two consecutive 16-bit registers. Contrary to S32_4321, the high-order word is returned in the second register, the high-order word in the first. In effect, the 4 bytes are returned in 2-1-4-3 format.
U32-2143	A 32-bit unsigned value returned in two consecutive 16-bit registers. Contrary to U32_4321, the high-order word is returned in the second register, the low-order word in the first. In effect, the 4 bytes are returned in 2-1-4-3 format.
S32-MFP or S32-M10k-4321	A 32-bit signed value returned in two consecutive 16-bit registers. The word stored in the first 16-bit register is multiplied by 10000 and added to the word stored in the second 16-bit register. Also known as 'signed Modulo 10000'.
U32-MFP or U32-M10k-4321	A 32-bit unsigned value returned in two consecutive 16-bit registers. The word stored in the first 16-bit register is multiplied by 10000 and added to the word stored in the second 16-bit register. Also known as 'unsigned Modulo 10000' or mod-10K.
S32-M10k-2143	A 32-bit signed value returned in two consecutive 16-bit registers. Contrary to S32_M10k_4321, the word stored in the second 16-bit register is multiplied by 10000 and added to the word stored in the first 16-bit register.
U32-M10k-2143	A 32-bit unsigned value returned in two consecutive 16-bit registers. Contrary to U32_M10k_4321, the word stored in the second 16-bit register is multiplied by 10000 and added to the word stored in the first 16-bit register.
S48-M10k-21-65	A 48-bit signed value returned in three consecutive 16-bit registers. $R3*10,000^2 + R2*10,000 + R1$, where R3 is the last register and R1 is the first register. Each registers range is -9,999 to +9,999
U48-M10k-21-65	A 48-bit unsigned value returned in three consecutive 16-bit registers. $R3*10,000^2 + R2*10,000 + R1$, where R3 is the last register and R1 is the first register. Each registers range is 0 to +9,999
S64-M10k-21-87	A 64-bit signed value returned in four consecutive 16-bit registers. $R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1$, where R4 is the last register and R1 is the first register. Each registers range is -9,999 to +9,999
U64-M10k-21-87	A 64-bit unsigned value returned in four consecutive 16-bit registers. $R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1$, where R4 is the last register and R1 is the first register. Each registers range is 0 to +9,999

Format Type	Description
S64-87-21	A 64-bit signed value returned in four consecutive 16-bit registers. The highest order word is returned in the first register, the lowest order word in the fourth. In effect, the 8 bytes are returned in 8-7-6-5-4-3-2-1 (or big ENDIAN) format.
U64-87-21	A 64-bit unsigned value returned in four consecutive 16-bit registers. The highest order word is returned in the first register, the lowest order word in the fourth. In effect, the 8 bytes are returned in 8-7-6-5-4-3-2-1 (or big ENDIAN) format.
S64-21-87	A 64-bit signed value returned in four consecutive 16-bit registers. The highest order word is returned in the fourth register, the lowest order word in the first. In effect, the 8 bytes are returned in 2-1-4-3-6-5-8-7 (or little ENDIAN) format.
U64-21-87	A 64-bit unsigned value returned in four consecutive 16-bit registers. The highest order word is returned in the fourth register, the lowest order word in the first. In effect, the 8 bytes are returned in 2-1-4-3-6-5-8-7 (or little ENDIAN) format.
S16-1-15	A 16-bit signed value. Bits 1 to 15 bits are unsigned data. If bit 16 is 0, the value is positive, if bit 16 is 1, the value is negative.
IEEEFloat or F32-4321	A 32-bit IEEE floating point value returned in two consecutive 16-bit registers. The high-order word is returned in the first register and the low order word in the second. In effect, the 4 bytes are returned in 4-3-2-1 format.
SwappedFloat or F32-2143	A 32-bit IEEE floating point value returned in two consecutive 16-bit registers. Contrary to F32_4321, the high-order word is returned in the second register and the low order word in the first. In effect, the 4 bytes are returned in 2-1-4-3 format.
MaskedBool or PackedBool	A 16-bit value that is interpreted according to the bit pattern described by the Mask attribute. Bits exposed by the mask can be read or written without affecting the value of other bits. Unmasked bits are interpreted as 0 on a read and are unaffected on a write. Undeclared bits of a mask are interpreted as unmasked (i.e., mask="0x7F" is interpreted as 0x007F). The value that is read or written is determined by using enumeration ordinals, described below. If used for read-only Boolean data, enumerations are not required. If no mask is specified, all bits are relevant.
PF-Nexus	A 16-bit unsigned value, range 0 to 3999, representing 3 decimal places of accuracy.
BCD (Binary Coded Decimal)	A number is expressed as a sequence of decimal digits and then each decimal digit is encoded as an 8-bit binary number. For example, decimal 92 is encoded as 00001001 00000010.

Format Type	Description
The following data types can have their length specified by the "Number of Registers to Request" setting.	
Packed BCD	A number is expressed as a sequence of decimal digits and then each decimal digit is encoded as a 4-bit binary number (nibble) For example, decimal 92 is encoded as 1001 0010.
ASCII	A sequence of bytes representing the ASCII character set. Each word stores two ASCII characters. Trailing spaces are removed.
ASCII-Reverse	Same as ASCII except every second character is in the reverse order.

Additional supported formats: U16-21, S16-21, U16-12, S16-12, U64-M10k-87-21, S64-M10k-87-21, U48-M10k-65-21, S48-M10k-65-21, U48-65-21, S48-65-21, U48-21-65, S48-21-65, InvertedMaskedBool, F64-87-21, F64-12-78, PF32, PF_ALT, U16-21-ARRAY, U16-12-ARRAY, DateTime4_UTC, DateTime4_LOCAL, DateTime4, DateTime3_UTC, DateTime3_LOCAL, DateTime3, DateTime_YMDhms_UTC, DateTime_YMDhms_LOCAL, DateTime_YMDhms, DateTime_IEC870_UTC, DateTime_IEC870_LOCAL, DateTime_IEC870, DateTime3_IEC870_UTC, DateTime3_IEC870_LOCAL, DateTime3_IEC870, DateTime3_MDYhms_UTC, DateTime3_MDYhms_LOCAL, DateTime3_MDYhms, DateTime4_MDYhms_UTC, DateTime4_MDYhms_LOCAL, DateTime4_MDYhms, DateTime2_s2000, DateTime3_s2000, DateTime4_shmMDY, DateTime6_smhDMY, DateTime7_YMDhms, DateTime8_MDYdowhmsc, DateTime6_MDYhmms, DateTime_NSX2_UTC, DateTime_NSX2_LOCAL, DateTime_NSX3_UTC, DateTime_NSX3_LOCAL, S64-M1K-87-21, U64-M1K-87-21.

Modbus Register Properties

The following register properties can be configured:

Name

The Modbus register name is an identifier for the register. It is used internally, but not displayed in the measurement selector user interfaces. The ION register label, to which the Modbus register is mapped, is used for display. If no ION register label is defined, then the ION register name is used for display.

For multi-circuit devices, such as the BCPM, the Modbus register name is used to identify the different circuits, with a special naming convention (example: `Real Energy@!!DeviceName!!_ 1PH_Ch1`). These special register names are then used for the creation of managed circuits, if that feature is configured for the device type in the Management Console.

> **NOTE**: The register name has a **maximum character limit of 50**, including spaces. This is important to consider when creating names for multi-circuit devices.

Constant Value vs. Mapped Value

To have an ION register linked to a measured value from a device, map this register to the appropriate Modbus register. To assign a constant value to an ION register, enter this value into the Value box for the ION register in the String Register Details pane. You must unmap a mapped

register first before you can edit the value of an ION register. To unmap a register, delete the ION Handle number for the mapped Modbus register, or delete the entire Modbus register row for the mapped Modbus register in the Register Map.

Disable Not Available Check

Use this field to skip the Not Available check for a register.

Enumeration

Enumeration is used to convert ordinals (position numbers) read from the device into the values these ordinals represent.

For example, a device may represent the Baud rate with a number from 0 to 3, where 0 = 19200 Baud and 3 = 115200 Baud. Using enumeration, the value 0 is converted to the string "19200 Baud", and the value 3 is converted to "115200 Baud". There are many applications for using enumerations when reading Modbus register data.

To create an enumeration:

1. In Device Type Editor, open the device type for which you want to define enumerations, and then browse to the **Enumerations** tab.

   > **NOTE**: This tab is hidden by default. To show the tab, check **View > Register Map > Enumerations**.

2. In Enumerations, insert a new row, using the context menu commands.

3. Enter a name in the **Name** field to identify this enumeration.

   For the example above, the name could be "Baud_Rate".

4. Enter a default value for the enumeration in the Default Enum Value field.

   For the example above, this could be "19200 Baud".

5. Click the Enum Info Option icon ⊞ in the name field for this enumeration to open the **Enum Info Option** table.

6. In Enum Info Option, insert new rows, using the context menu commands and enter the ordinals into the **Ordinal** fields and the corresponding values into the **Value** fields.

   For the example above, ordinal 0 has a value of 19200 Baud, ordinal 1 a value of 38400 Baud, ordinal 2 a value of 57600 Baud, and ordinal 3 a value of 115200 Baud.

7. Click the ⊠ icon in the op left corner of the Enum Info Option grid to return to the **Enumerations** table.

8. Browse to the **Register Map** table and enter the enumeration name created in step 3 into the **Enum Name** field for the registers to which it applies.

Format

This data format is used for decoding the values returned from the meter. The selected data format must match the format delivered by the device. See Modbus Data Formats for the supported formats.

ION Handle

The ION handle is the reference ID used by WinPM.Net to identify the ION register that is mapped to the Modbus register. The ION handle is automatically assigned when you map a Modbus register from the Register Map to the ION register in the ION Tree.

Mask

The mask is used to read individual bit values from a Modbus register value. It is applied to the register data using an AND logic. For example, if the mask "0x4" is applied, then the register value displayed is only nonzero if the 3rd least significant bit in the register is a 1. You can apply masks to both numeric and Boolean data types.

> **NOTE**: For integer values, the bit order of the mask corresponds to the bit order of the actual data and NOT the order in which the bytes representing the data are returned over Modbus. The high-order nibble of a 32-bit value is selected as '0xf0000000' regardless of whether the format is U32_4321 or U32_2143. It should also be noted that the resulting value of a masked integer is NOT bit shifted. A mask of '0xf000' applied to a 16-bit value of '0x1234' results in '0x1000' (4096 decimal) and NOT 1.

Modbus Address

This is the physical address of the Modbus register. You can find this address number in the device documentation. Enter the number in either decimal or hexadecimal format. The address range depends on the register type as follows:

COIL STATUS = 0xxxx
INPUT STATUS = 1xxxx
INPUT REGISTER = 3xxxx
HOLDING REGISTER = 4xxxx

> **NOTE**: When entered in hexadecimal format, the value is only the register offset and does not include the Register Type identification.

Not Available Value

If the Modbus register has a control signal value to indicate when the register is not available, then you can enter this value in this field. The value must match the control signal value that is received from the device. This allows the user interfaces that display data for this register in WinPM.Net to indicate when the value is not available.

Number of Registers

This option is only available if the data type is set to ASCII, ASCII-Reverse or Packed BCD. It allows you to specify the number of registers to retrieve from the device on a request.

Request Type

This tells the system whether the register is for read (R) or write (W). Other options are read/write (RW), and command write (CW).

Scale, Multiplier, Offset

The scale, multiplier and offset can be set to any decimal value within the range of -3.402823466 E + 38 to 3.402823466 E + 38. These values are applied to the measured value returned from the device in the following way:

ION value = (Modbus value - offset) * multiplier / scale

Instead of setting these values to a constant, you can also map them to ION registers that hold these values. This is useful if the value of the ION registers can be read from the device. It is also useful if you want to use the same scale, multiplier, or offset for multiple registers. In this case you can map all of the registers to the same ION registers, and set their values to constants. You can only map to registers already created in the Device Type Editor.

To map the scale, multiplier, or offset to an ION register, enter the ION handle value for this register, in hex, into the Scale, Multiplier, or Offset fields in the Modbus Register Map.

Setup Reg

Use this field to indicate that a register is a setup register on the device. Setup registers are read at a lower polling rate than regular data registers, improving the overall performance of the device driver.

Write Value

This only applies to pulse registers. Use this field to specify the value that is written to the Modbus register on the device when the ION register is pulsed.

Measurement Tree

The Measurement Tree organizes the device measurements into a structure that is recognized by WinPM.Net and that determines how you access these measurements in the software. The Measurement Tree consists of managers, modules, and registers.

Managers

Managers are at the top level of the tree. Managers group together modules of the same type. For example, a manager called **Maximum Readings** could contain the modules **Max Current**, **Max Frequency** and **Max Voltage**. You are free to choose a name for a manager that best meets your needs. Choose a name that is relevant to the type of modules it contains. You can create up to 127 managers.

Modules

Modules are contained inside managers. Modules group together related registers. For example, a module called **Max Current** could contain the registers **Max Current A**, **Max Current B**, **Max Current C**. You are free to choose a name for a module that best meets your needs. Choose a name that is relevant to the type of registers it contains. You can create up to 4095 modules inside a given manager.

Registers

Registers are contained inside modules. (ION) Registers are linked to the Modbus registers and hold the measured values. You can map registers to a Modbus address, then use WinPM.Net to read and write data to this Modbus device. You can create up to 255 registers per module.

Factory registers

You cannot add or remove Factory registers, but you can edit them. All Factory registers are initially set to a default constant value. You can change these constant values, or map the registers to a valid Modbus address.

OPC Tags

A complete OPC tag contains the elements shown in the following example:

Server name and address	Channel name	Device name	Tag name

`Opcda://Server/OpcDa.1.Channel_1.Device_1.Powers.Real_Power`
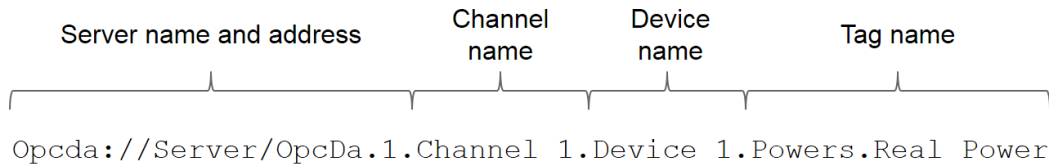
Manual Data Editor user interface

Data editor user interface (UI)

1	**Load New Data.** Click Load New Data to open the Load Data dialog box. In Load Data you select the sources, measurements, time range, and display options for the data that is loaded into the data editor. Load Data opens automatically when you start Manual Data Editor. See Load Data UI for more information.
2	**Locate Changes.** Use Locate Changes to find and step through any changes you made to the data loaded into the data editor, before you apply those changes. After you apply the changes, they become part of the data set in the database and Manual Data Editor no longer identifies them as changes.

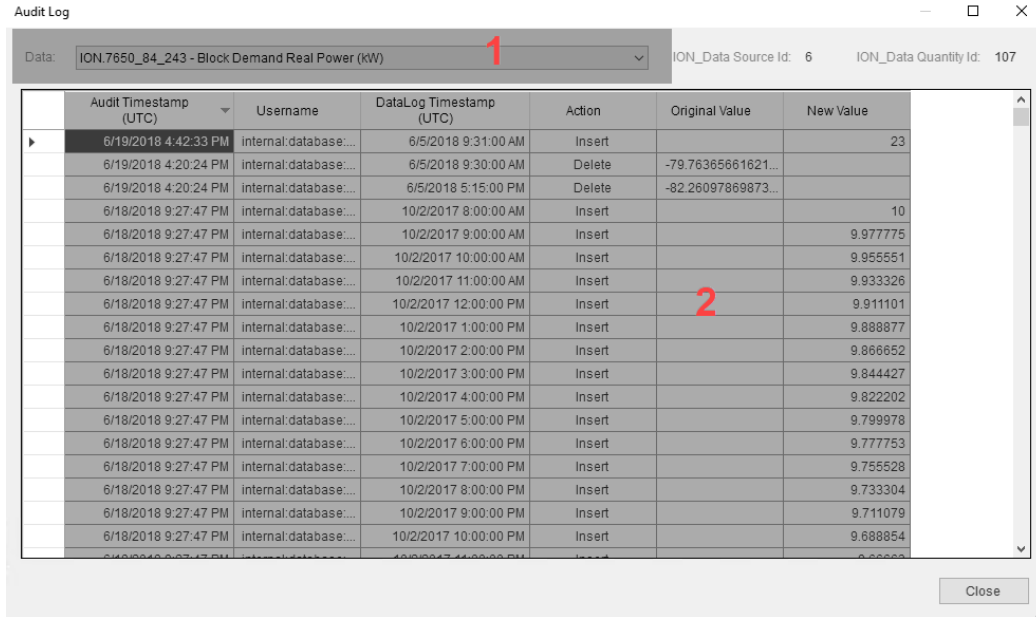| 3	**Locate Issues.** Use Locate Issues to find and step through any issues identified by Manual Data Editor in the data loaded into the data editor.
4	**View Issue Report.** Use View Issue Report to see which data issues were identified by Manual Data Editor in the data loaded into the data editor.
5	**Insert Timestamp.** Insert a new data record in the data loaded into the data editor with a timestamp you can specify. The timestamp must be in the time range of the data that is loaded into the editor. The new record has <no data> data values. When adding new data records, keep in mind that a historical data log timestamp marks the end of the logging interval. For example, a data log with a 15 minute logging interval and timestamp of 17:00 represents data for the time interval of 16:45 - 17:00.
6	**Smart Gap Fill.** Use Smart Gap Fill to fill in missing data in the data set loaded into the data editor. You can select a range of data that you want to fill the gaps in or you can fill all gaps in the entire loaded data range. When selecting a data range in the editor, only select the data values, not the timestamps. See Smart Gap Fill UI for more information.
7	**Data grid.** The data grid shows the sources/measurements that you specified in the Load Data dialog box. Use the grid and the functions in the data editor to edit the data as needed. **TIP**: Move the pointer over the header of a data column to see a tooltip with additional information.
8	**Context menu.** Right-click a data cell in the data grid to open the context menu. The Context menu gives you access to the commands and functions for editing the data loaded into the data editor.
9	**Color legend for issues highlight** The data editor uses different color highlights to mark data issues. This legend tells you which colors are used for which type of issues.
10	**View Log** Click View Log to open the Manual Data Editor historical audit log. See Audit Log UI for more information.
11	**Undo All Changes** You can undo changes that have not been applied yet. After changes have been applied to the database, they are permanent and can only be modified be manually editing the data again in the Manual Data Editor.
12	**Apply Changes** After you have completed your data edits, click Apply Changes to write the changes to the database. After changes have been applied to the database, they are permanent and can only be modified be manually editing the data again in the Manual Data Editor.

Load Data UI

1	**Available Sources.** Select the source for which want to load the data.
2	**Available Measurements.** Select the measurement for which you want to load data. You have the option to only see those measurements that have logged data for the selected source in the drop-down control, or see all measurements that are defined in the software.
3	**Add Data to Selection** Click Add Data to Selection to add the source/measurement you selected, to the list of data to be loaded into the data editor. You can add multiple measurements from the same source, and measurements from different sources to the list.
4	**Selected Pairs** This shows the list of selected source/measurement pairs that are to be loaded into the data editor.
5	**Edit commands for the Selected Pairs list** Use **Clear All** to remove all selected list items. Use **Remove Selected** to remove items that you selected from the list.

6	**Create new Measurement** Use this to create a new measurement in the software. This function is needed if you want to add data that was manually collected for a measurement that does not exist in the software. For example, if you wanted to add a daily count of cars parked in a parking lot, you could define a measurement called Number of Cars, which doesn't exist in the software by default. Then you can select this measurement and a source in Manual Data Editor and add data records with the daily counts. **NOTE**: Check the list of available measurements to see if one exists for your application before creating a new measurement.
7	**Preset** Save and load settings for the Load Data dialog box. This is a way to simplify the data selection for data sets that you frequently access. Set all the settings in Load Data for the data sets you want and then save the settings to an external file using **Save Preset**. Load a pre-defined set of settings into Load Data using **Load Preset**.
8	**Time Range** Select the time range for which you want to load the data from the sources/measurements.
9	**Timezone** Set the timezone for the display of the configurable timestamp in the data editor. The data editor shows two timestamps for each data record, one timestamp in UTC and one in a configurable timezone.
10	**Timestamp Format** Set the format for the timestamp display in the data editor.
11	**Number Format** Set the number of decimals that are displayed for the logged data in the data editor. **NOTE**: This setting only affects the display of the data in the data editor and the accuracy to which you can enter new data values. It does not affect data that is loaded into the editor and then applied to the database without editing the values.
12	**Is Interval Data** Set the expected logging interval for the data. This setting ensures that there is a data row in the editor for every expected log entry. If there is no data record for an expected entry in the database, then the Manual Data Editor inserts a timestamped row with a \<no data\> data values in the editor. This makes it easier to identify missing records and correct them. **NOTE**: The data editor shows all logged records in the selected time range, regardless of the Interval Data settings. No records are hidden or filtered out.
13	**Load Data** Click Load Data to open the selected sources/measurements data in the data editor.

Audit Log UI

	Data.
1	Select the source/measurement for which want to see the audit data. The available options include all sources/measurements that have been loaded into the data editor.
	Audit log.
2	View the change history for edited data records including timestamp, username, action, original value, new value. You can use the information in the audit log to reset data values to their original value if they were changed accidentally or incorrectly.

Smart Gap Fill UI
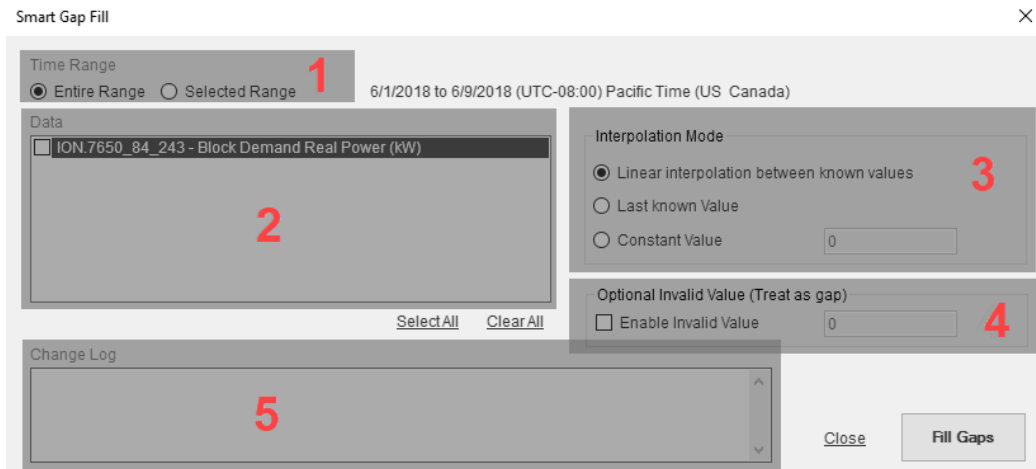
	Time Range.
1	Select the range of data that you want to fill the gaps in or you can fill all gaps in the entire loaded data range. When selecting a data range in the editor, only select the data values, not the timestamps.

2	**Data.** This is a list of all the sources/measurements that are loaded into the editor. Select the source/measurement pairs for which you want to fill gaps.		
3	**Interpolation Mode** Choose which algorithm is used to fill the gaps. You have the following choices: 	Linear interpolation between known values.	Gaps are filled with the average of the last recorded value before the gaps and the first recorded value after the gaps. All consecutive gaps between the known values are filled with the same value.
Last known Value	Gaps are filled with the last recorded value before the gaps. All consecutive gaps after the known value are filled with the same value.
Constant Value	Gaps are filled with the value that you specify in the value box.			
4	**Optional Invalid Value** You can define a data value that is treated by the gap fill algorithm as if it was a gap. For example, if you define 0 (zero) as the Invalid Value, then any data record in the Smart Gap Fill range that has a value of 0 is treated as if it was a gap.		
5	**Change Log** The Change Log shows how many gaps were filled, for each of the selected source/measurement pairs, after you click **Fill Gaps**. **NOTE**: This is different from the Manual Data Editor audit log. The audit log records changes that are applied to the data in the database. The Change Log shows which gaps where filled in the data editor.		

WinPM.Net 10.0

User Guide

For more information visit
www.usa.siemens.com/access