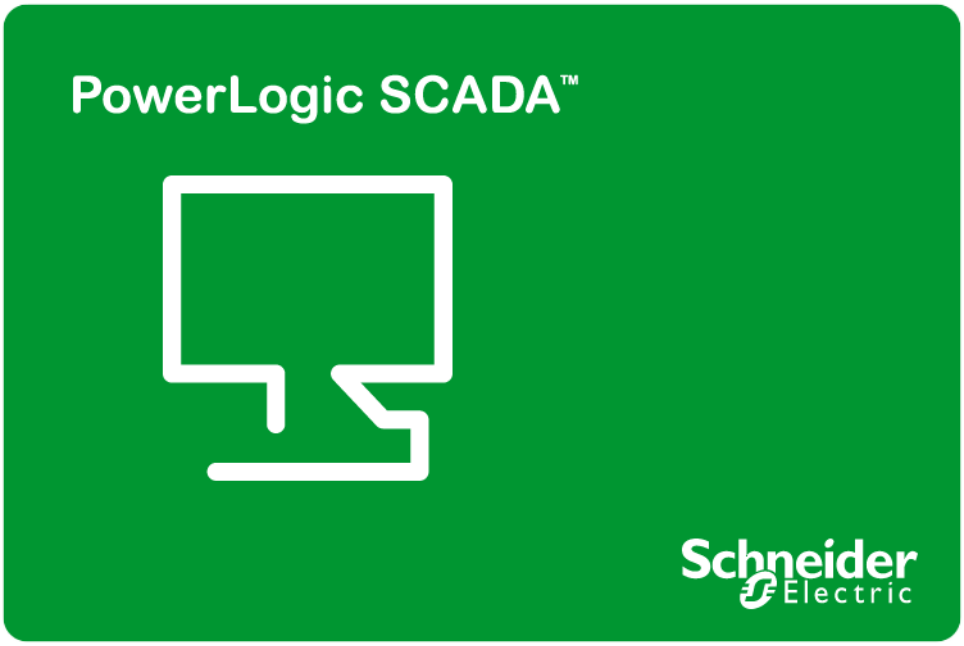
# PowerLogic™ SCADA

## System integrator's manual
## Version 7.20

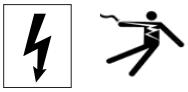
63220-100-200E1
06/2011




Schneider Electric

## HAZARD CATEGORIES AND SPECIAL SYMBOLS

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 an imminently hazardous situation which, if not avoided, **will result** in death or serious injury.

**⚠ WARNING**

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

**⚠CAUTION**

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

**CAUTION**

**CAUTION**, used without the safety alert symbol, indicates a potentially hazardous situation which, if not avoided, **can result** in property damage.

NOTE: Provides additional information to clarify or simplify a procedure.

## PLEASE NOTE

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

# Contents



© 2011 Schneider Electric All Rights Reserved 

© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved 

© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved 

© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved 

© 2011 Schneider Electric All Rights Reserved

# Introduction

Welcome to the help file for PowerLogic™ SCADA version 7.20. To navigate through the file, you can use the search options on the left. Additionally, you can access the PowerLogic SCADA help files for PowerLogic SCADA-specific information.

In this section, you will find these topics:



© 2011 Schneider Electric All Rights Reserved

# PowerLogic SCADA—A Complete Solution

The PowerLogic SCADA system provides a complete solution for the monitoring and control of any electrical distribution network. Using PowerLogic SCADA as the engine, PowerLogic SCADA includes additional tools that users will use to acquire data from various devices, and then to apply the data for HMI viewing, alarming, trending, and reporting.

In this section, you will find these topics:



## Assumptions

The person who installs PowerLogic SCADA will have received training in the entire PowerLogic SCADA product and will understand the application's basic functions.

## What is SCADA?

SCADA (Supervisory Control and Data Acquisition) is a system that collects data from various points, both local and remote, and then stores the data at a central location. The system also can control equipment at an installation.

## What is PowerLogic SCADA?

The PowerLogic SCADA product includes several tools that use the PowerLogic SCADA engine to collect information from all PowerLogic devices:

- In the Profile Editor, you create, configure, and modify profiles for device types that use the same tags (for example, to set up alarms and trends).
- In the Profile Wizard, you add and modify devices in the PowerLogic SCADA system.
- In the Project Color Change Tool, you set colours for the different states of circuit breakers and busbars. These colours display on the final runtime environment page.
- In the CitectSCADA Graphics Builder (design time mode), you create one-line drawings, into which you can incorporate objects (genies). Genies are versatile objects that include variables or other expressions that are specified when you view the drawing.
- In the CitectSCADA Graphics Builder (runtime mode), users can view the final one-line drawings, including alarms, events, and history data. With the appropriate degree of password-controlled authority, users can also perform advanced tasks, such as changing alarm setpoints and racking devices in and out.

## Preparing for PowerLogic SCADA

For a full discussion about preparation for a PowerLogic SCADA project, see Planning a Project in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help). This section of the help discusses physical layout; requirements such as architecture and security; and project design.


© 2011 Schneider Electric All Rights Reserved

# Components of a Project

For detailed information about the components that make up a PowerLogic SCADA project, see Components of a project in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help). The project components include graphic components, tags, alarms, system components, communications components, I/O Server components and Cicode/CitectVBA.

## Configuration Tools

Configuration tools consist of:

**Profile Editor**: Use this tool to select tags to be used by device types (tags must be consistent with IEC 61850 naming conventions), create device profiles for individual devices, and create projects that include the device profiles to be used in a single installation. You can specify real-time tags, PC-based alarm tags, onboard alarm tags, trend tags, and reset tags to be generated for this device.

**Profile Wizard**: Using this wizard, you will import device profile information from the Profile Editor into a PowerLogic SCADA project. This tool is simply a means of moving device profile information into the PowerLogic SCADA project and converting it into formats that PowerLogic SCADA can use.

**Citect Explorer**: Use Citect Explorer for basic navigation. From here, you also choose the active project.

**Citect Project Editor**: Use the Project Editor for entering database-type information, such as adding clusters and servers, creating new users, and editing tags within PowerLogic SCADA projects.

**Graphics Builder, Design-Time environment**: Use the Citect Graphics Builder to create one-line drawings that users can view in the runtime environment. These drawings are populated with interactive objects that are generated by genies. You can also use the graphics tool to set up system alarms and trends.

**Project Color Change Tool**: This tool is part of the Project Editor. Use it to change colours for the states of busbar and circuit breaker genies used in graphics pages.

## Runtime Environment

The runtime environment is where the end user views system information. This environment includes (from the information added in the design-time page) one-line pages with interactive objects, alarm and event pages, and analysis pages (trends and waveforms).

## Supported Device Types and Protocols

When you install PowerLogic SCADA, you are prompted to choose the drivers that you will use. A certain number of generic drivers are installed by default (including PowerLogic device types), and you are not prompted for them. Device types and protocols supported in PowerLogic SCADA are:

- Generic MODBUS (includes BCPM and any device, such as a PLC or UPS, that communicates via MODBUS). When adding a controllable device in the Profile Editor, such as a circuit breaker, use the “Controllable Device” driver; otherwise, use the “Generic Power Device” driver.
- Sepam 20, 40, and 80 Range, 2000
- MicroLogic 5P and 6P, A, H
- CM4000 series

© 2011 Schneider Electric All Rights Reserved

- PM800 series
- PM700 series
- ION 7650
- IEC 61850 protocol devices
- IEC 870-5-104 (requires extra purchase)
- DNP3


© 2011 Schneider Electric All Rights Reserved

# System Requirements and Installation Instructions

See the *PowerLogic SCADA Installation and Configuration Guide* for a complete discussion of system requirements and installation instructions. A PDF copy is on the installation disk.

## Uninstalling and Reinstalling PowerLogic SCADA

Use Add/Remove Programs in the Control Panel to uninstall these PowerLogic SCADA programs:

- PowerLogic SCADA 7.20 (if you uninstall this, you also uninstall the Profile Editor and the Project Color Change Tool)
- PowerLogic SCADA Profile Editor 7.20
- PowerLogic SCADA Project Color Change Tool
- Any additional PowerLogic SCADA programs, such as the WebServer, that you installed

If you uninstall PowerLogic SCADA after you have already created projects, the project data will not be deleted. It is in [Project Drive]\Documents and Settings\ All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20 (for Vista operating systems, it is [Project Drive]\ProgramData\Schneider Electric\PowerLogic\PowerLogic SCADA 7.20). The first time you launch the application after you re-install it, it will locate the project data and re-link it.

© 2011 Schneider Electric All Rights Reserved

# Quick Start: Typical Workflow

The following is a brief description of the steps you will take when you create a (Modbus or non-61850) PowerLogic SCADA system. Also, see *Typical Workflow Illustration* on page 22 for a flow chart that illustrates the Profile Editor workflow.

1. **Identify the equipment and devices to be used in the system**: List all of the device types that you will need.
2. **Create a system architecture drawing**: Indicate how all devices will interrelate.
3. **Prepare the network addressing scheme**: Define the Ethernet IP address for each gateway/PLC/CM4000 and the MODBUS slave address devices.
4. **Install the hardware**: Install the devices and equipment for the system. Create all of the communications connections that are necessary. Make sure that devices are set up for the onboard alarms and waveform captures that you want to view from graphics pages.
5. **Set up device type categories and units/conversions in the Profile Editor**: Use the Setup Device Type Categories screen to add categories (an optional feature that will make it easier to sort device types in the Profile Wizard). Also use the Profile Editor to add unit conversions (e.g., watts/megawatts) that are used in individual installations. Most typical conversions have already been added to PowerLogic SCADA.
6. **Add/edit device types in the Profile Editor**: The device type includes all of the details that define a device: real-time tags, alarm tags, trend tags, resets, and controls. Use the Define Device Type Tags tab to add and edit device types, create custom tags, and to assign tags to device types. Although the device type includes all of the information we know about a device, the device profile (see step 8) includes only the specific tags that are needed for that device in a customer's installation.
7. **(optional) Edit tag addresses in the Profile Editor**: As needed, add or edit the address attributes of individual tags for a single device type. This includes such items as the number of registers used by the tag, bitmasking, and logic code.
8. **Create device profiles in the Profile Editor**: The profile is a subset of the device type's functionality. Use the Create Device Profile tab to create a device profile; select only the tags required for a customer. On this tab and the related Add/Edit Device Profile screen, you also specify tags to be used for onboard waveform captures, PC-based alarms, and trends.
9. **Create and configure a project in Citect Explorer**: From Citect Explorer, add a new project. From the Project Editor page, add at least one cluster and network address, and the servers that will be used (alarm, report, I/O, and trend). Set 'Publish Alarm properties' to TRUE when setting up the alarm server (expand the screen by pressing F2). This must be set; otherwise you will not be able to edit while in the setpoints page.
10. **Create a corresponding project in the Profile Editor**: In the Set Up Projects tab, select the device profiles for the devices that are included in the project created in the previous step. Add the project to the Profile Editor, using the same name you created in the previous step. To view the list of projects created in PowerLogic SCADA, click the Display Projects button (on the Add/Edit Project screen) :

    Each profile will include all of the device attributes that you have set up in earlier steps.
11. **Export devices from the Profile Editor**: From the Set Up Projects tab, choose a single project and click Export Project to convert device tags into a format usable


© 2011 Schneider Electric All Rights Reserved

within PowerLogic SCADA, and to move them to the location where they can be imported into PowerLogic SCADA.

12. **Use the Profile Wizard to set up clusters and add devices to PowerLogic SCADA**: Use the Profile Wizard to set up clusters (created in step 9). See *Adding a Device to a PowerLogic SCADA Project* on page 97 for additional information.

    Use the Profile Wizard to add the exported devices, one at a time; or use the Automation Interface (*Adding Multiple Devices through the Automation Interface* on page 107)to add all of the devices from one page (example found in: Program Files\Schneider Electric\PowerLogic SCADA 7.20\Bin*Example Automation.xls*). You will need device networking information for this step.

13. **Compile and correct errors in PowerLogic SCADA**: Note: It is always a good idea to pack before you compile (Project Editor > File > Pack). To compile the project, click Compile, located at the top of the Project Editor:

    Then correct any errors and note any warnings.

14. **Create graphics in the Citect Graphics Builder**: Use the Graphics Builder (the design time environment) to create one-line diagrams that incorporate icons and objects for the user to read information from the system. You will use this feature to add genies that, when launched from the runtime environment, provide real-time system information.

15. **Use the Menu Configuration tool to set up the appearance of the runtime environment**. The Menu Configuration tool creates/edits the pagemenu.dbf file, which controlsthe appearance of tabs and menus for a specific project in the runtime environment. Access it via the Citect Project Editor (System > Menu Configuration). You can copy and paste the pagemenu.dbf file from the PLS_ Example project to your own project; or you can configure your own template from scratch.

16. **Add at least one user in the Citect Project Editor**. From the Users window (System > Users), add at least one user to the project.

17. **Set colours for busbars and circuit breakers in the Project Color Change Tool (Citect Project Editor)**: Click Tools > Project Color Change Tool to launch. Use the Color Change Tool to add unique colours for different states of busbars and circuit breakers. You can also add a second colour if you want the object to flash between the two colours when it is in a particular state. (Each time you add a genie, after the system is set up, you need to run the Project Color Change Tool again to ensure that the colours are assigned to the new genie.)

18. **Run the Computer Setup Wizard**. This wizard, located in Project Editor > Tools > Computer Setup Wizard, customizes your computer for use with PowerLogic SCADA. To run the wizard, click:

    NOTE: When running the Computer Setup Wizard, do not change the startup page (leave the ini settings as <default.>) Changing the startup page at this point will result in navigation problems.

19. **Compile (File > Compile), then run the project**: Compile the project:

    Then correct any errors and note any warnings. Run the project:

© 2011 Schneider Electric All Rights Reserved

Each time you add genies to a project, you need to associate them with devices, and then compile and run again.

20. **Test the system**: Verify that the runtime environment objects all operate correctly, that the system is recording data and that all alarms and waveforms function.


© 2011 Schneider Electric All Rights Reserved

# IEC 61850 System Setup Workflow

These are the basic steps you need to follow to set up an IEC 61850 device in your PowerLogic SCADA project.

1. List all of the SCL files (ICD, CID) for the IEC 61850 devices in your installation. ICD files are preferred. Pay special attention to data concentrated devices (for example, the G3200 with multiple devices communicating through it; see *G3200 Device Setup* on page 62).
2. Import the first ICD file into the Profile Editor (see *Importing SCL Files* on page 76).
   a. Create the device type.
   b. Match or verify tags for PowerLogic SCADA.
   c. Complete the import.
3. Create a device profile for the IEC 61850 device type (see *Adding an IEC61850 Device to a PowerLogic SCADA Project* on page 103).
   a. If needed, add/edit datasets and report control blocks (see *Editing IEC 61850 Datasets* on page 57 and *Editing IEC 61850 Report Control Blocks* on page 58).
   b. Select the appropriate tags for PowerLogic SCADA to monitor for this device.
4. Repeat steps 2 and 3 for additional ICD files.
5. Create a Profile Editor project, adding the device profiles. Configure as needed.
6. Export to PowerLogic SCADA, and to SCL.
   — PowerLogic SCADA creates the equipment.profiles file for the Profile Wizard or Automation Interface.
   — SCL will create an IID file for the profile. If newly added datasets and/or report control blocks are to be used, this IID file is required for step 7. Otherwise, you can use the original ICD file.
7. Use the appropriate IEC 61850 configuration tool for the device to configure a CID file from the ICD/IID file. Then download it to the device.
8. Create the PowerLogic SCADA project.
   a. From within PowerLogic SCADA, add a new project.
   b. Add the appropriate clusters, networks, and servers.
9. Using the Profile Wizard or Automation Interface, add your devices to PowerLogic SCADA.

   When you are prompted for the SCL file, use the CID file you created in step 7.

   For more information, see *Adding an IEC61850 Device to a PowerLogic SCADA Project* on page 103.
10. Compile and run the project.

© 2011 Schneider Electric All Rights Reserved

# Getting More Information

For additional information, use one of these options:

*Manuals* on page 18

*Help files* on page 18

*Customer support* on page 18

## Manuals

In addition to this *System Integrator's Guide*, there are these additional documents that may provide helpful information:

- **PowerLogic Installation and Configuration Guide**: a PDF copy is on the installation disk
- **Release Notes**: located on the installation disk, this file includes information specific to this release of the product
- **Readme file**: located on the installation disk, this file also includes late-breaking information about this release

## Help files

In addition to the help file released with PowerLogic SCADA, there are several PowerLogic SCADA-specific help files. They are located in the PowerLogic SCADA Bin folder: \Schneider Electric\PowerLogic SCADA 7.20\Bin).

## Customer support

Americas: +1 250 544 3010 (24 hours, Monday-Friday)

Asia/Pacific: +852 2579 9568

Europe/Africa: +33 4 76 60 62 35

e-mail: Global-PMC-Tech-Support@Schneider-Electric.com

web support: www.powerlogic.com


© 2011 Schneider Electric All Rights Reserved

# The Profile Editor

This section describes how to use the Profile Editor to create device type tags, and then to use these tags as building blocks for device types. Secondly, there are instructions for creating device profiles for unique devices. Finally, there are instructions for creating projects for each installation or customer.

In this section, you will find these topics:



© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved

# Overview of the Profile Editor

The Profile Editor is a multiple-screen application that allows you to perform four basic tasks: create device types, create device profiles, and set up projects.

Use the **Define Device Type Tags** tab and its screens to add and edit information for real-time, onboard alarm, control and reset tags and to create and edit device types. See *Defining Device Types and Tags* on page 33 for complete instructions.

Use the **Create Device Profiles** tab and its screens to add and edit individual profiles for specific devices. A device profile is a subset of the possible variable tags, alarm tags, and trend tags for a particular device type. See *Creating Device Profiles* on page 51 for complete instructions.

Use the Set Up **Project** tab and its screens to bring together all of the system attributes for a single customer or installation. For example, the customer installation will include a certain combination of device profiles (depending on the devices installed at the site). The project allows a specific unit template to be applied, converting units (such as watts) into units used by the customer (such as megawatts). This causes tags to display in the converted format. Projects also allow you to rename tags to suit a customer's needs (for example, Current A could be renamed to Current Phase A). See *Setting Up Projects in the Profile Editor* on page 65 for complete instructions.

PowerLogic SCADA uses the IEC61850 tag-naming convention to create tags that measure device quantities. Although most of the tags you will use are already entered into the system, you can add custom tags. For more information, see *Understanding Tags* on page 78.

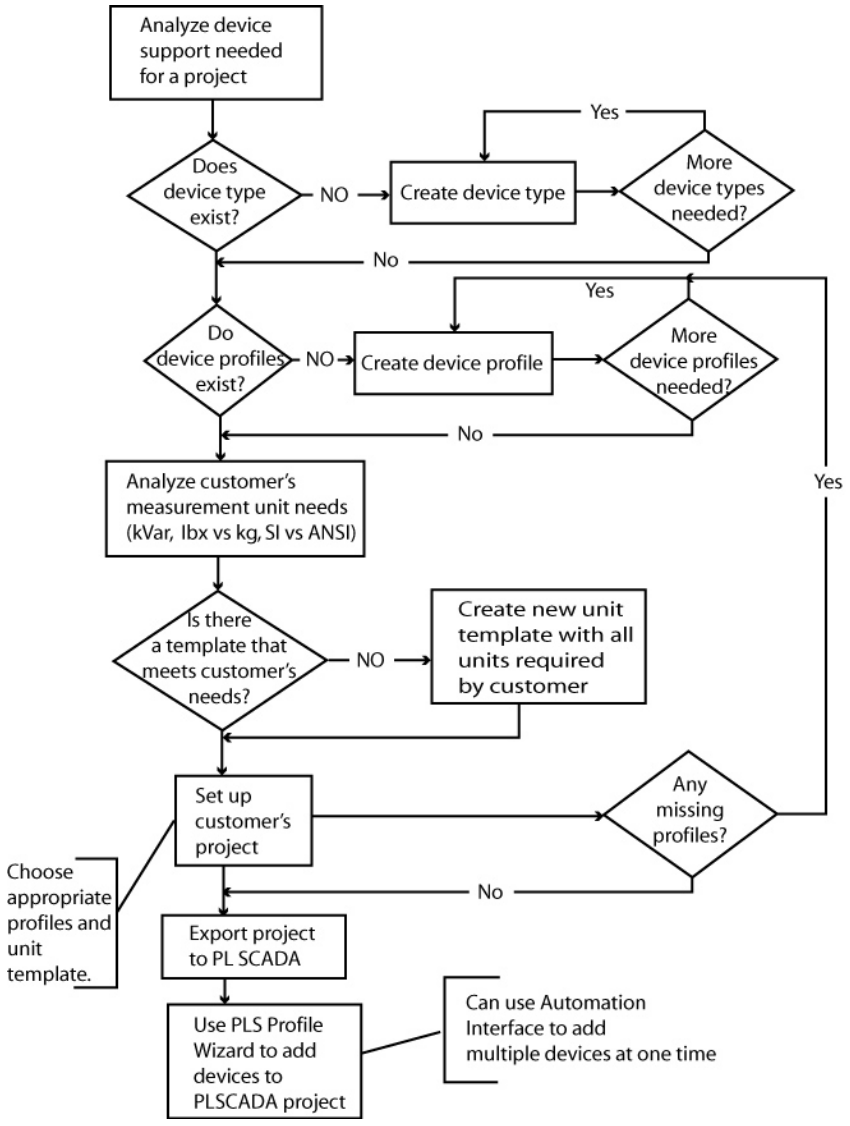
© 2011 Schneider Electric All Rights Reserved

# Typical Workflow Illustration

The following four flow charts illustrate the use of the Profile Editor. The first illustration provides an overview, and the following illustrations show:

- creating/editing a device type
- creating/editing a device profile
- creating/editing unit templates

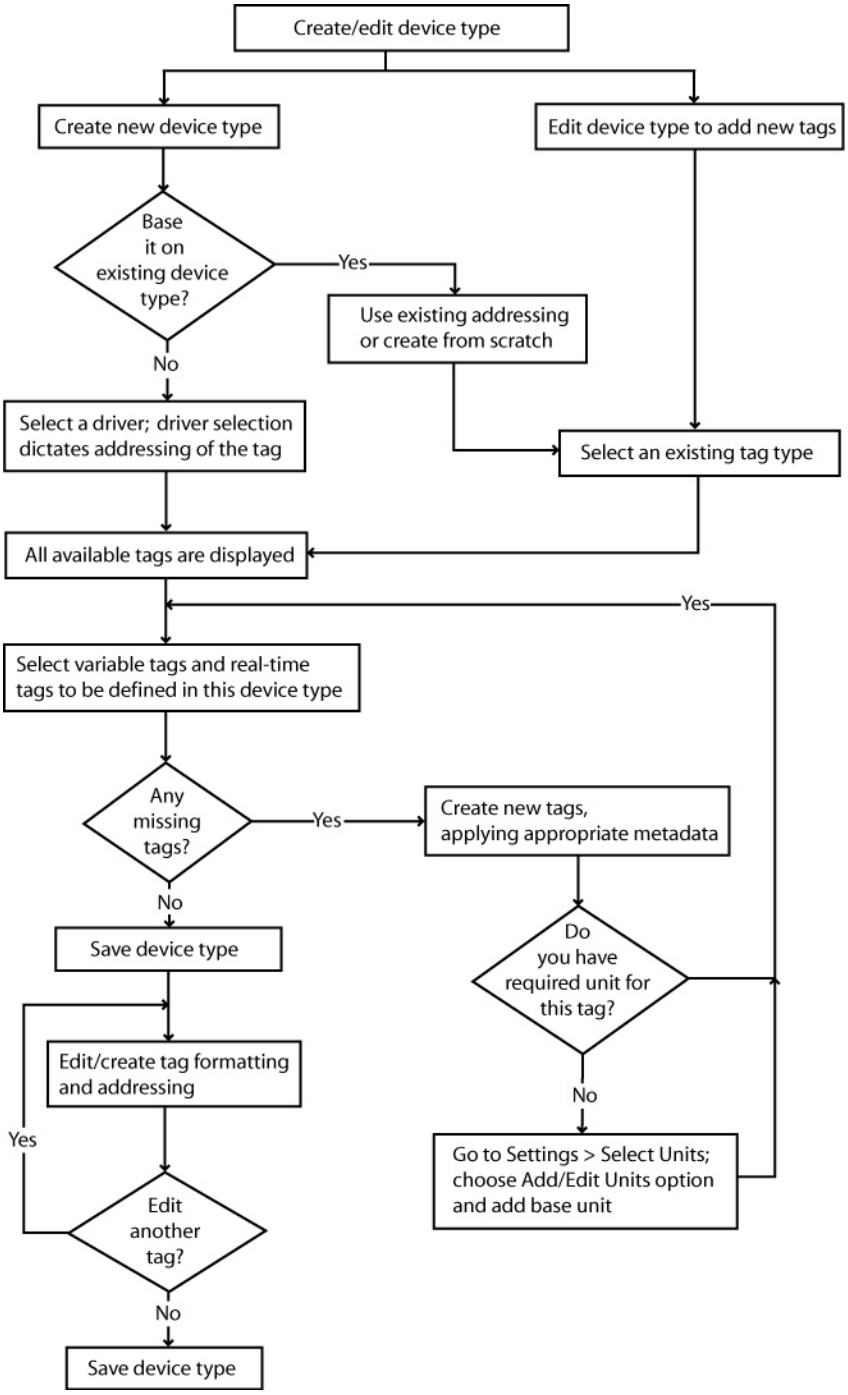
## Workflow Overview





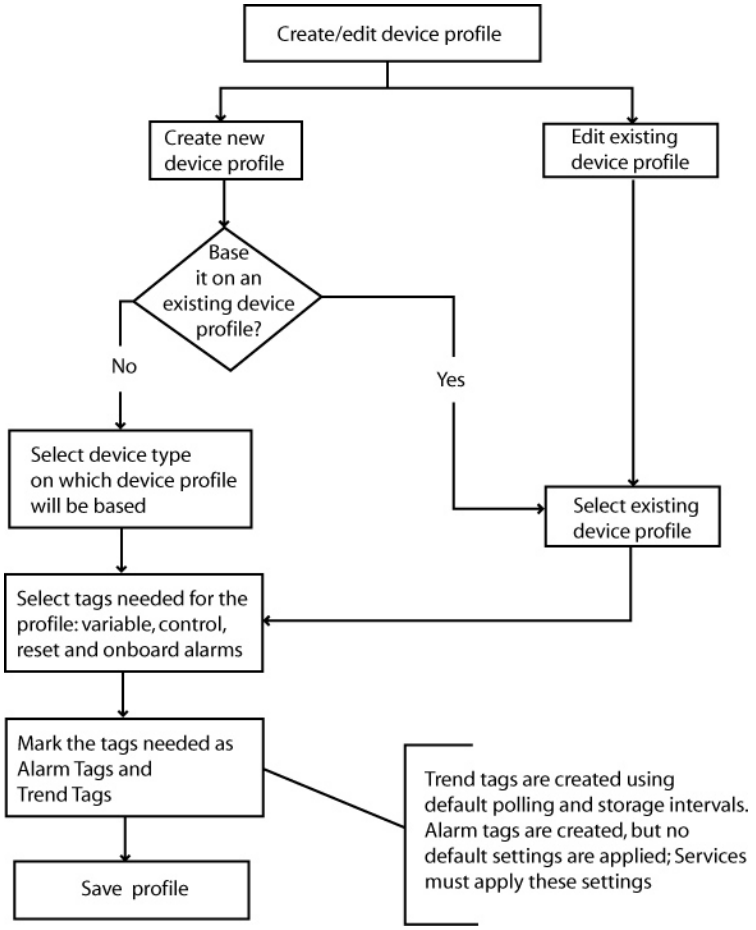
© 2011 Schneider Electric All Rights Reserved

## Create/Edit Device Type




© 2011 Schneider Electric All Rights Reserved

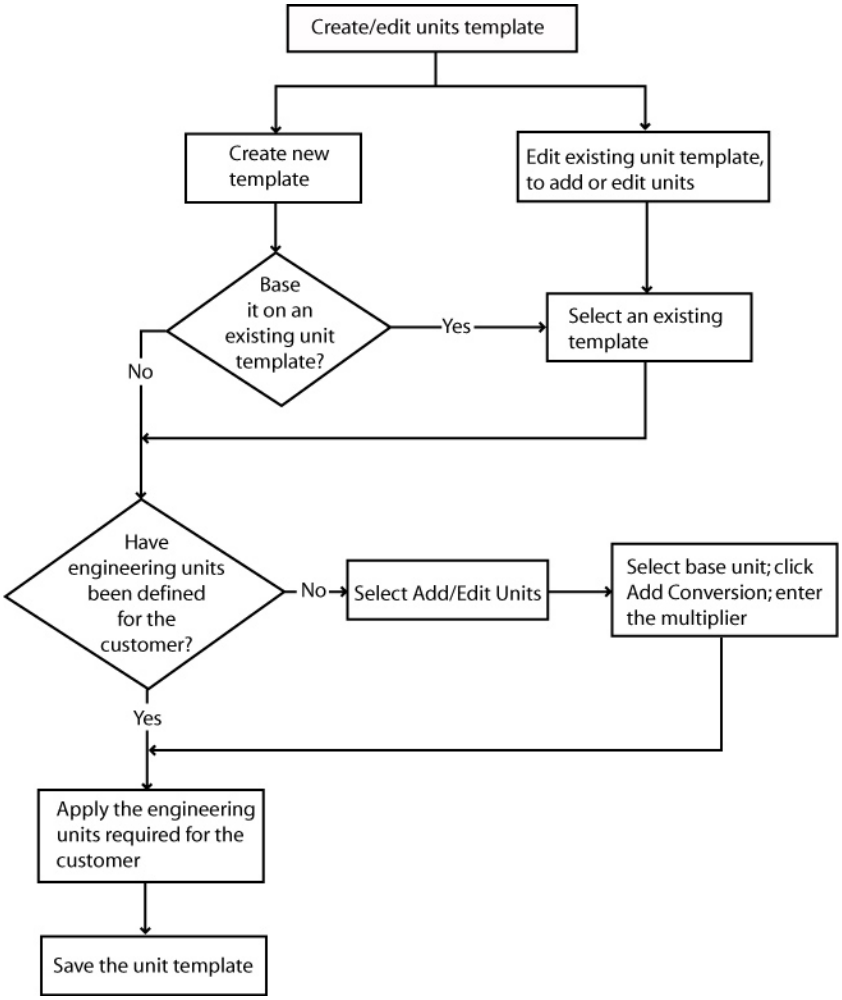
## Create/Edit Device Profile





© 2011 Schneider Electric All Rights Reserved

## Create/Edit Unit Templates




© 2011 Schneider Electric All Rights Reserved

# Launching the Profile Editor

To launch the Profile Editor, click Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > Config Tools > Profile Editor. The Profile Editor screen displays with the Define Device Tags tab selected. There are two other tabs, used to create device type profiles and projects.

### Locked and Custom Icons

Two icons may appear to the right of the Add/Edit button on some screens: the locked icon and the custom icon.

**The Locked Icon**  :

This icon indicates that the selected file (e.g., device type, profile, or project) cannot be edited. All standard device types (for example, Circuit Monitor 4000, MicroLogic Type P, Power Meter 800) are automatically locked; they cannot be unlocked.

To lock a device type that you create, follow these steps:

1. From the Define Device Type Tags screen, click Add/Edit.
2. On the Add/Edit Device Type screen, you can choose Create New, Create From, or Edit Existing.
3. Enter the new device information, or select the device to be edited.
4. To lock the device, check the box named Lock This Device Type.
5. Click Save & Exit to save the “lock” and exit the screen.

After you lock a device type, you cannot unlock it. However, you can restore it in this manner: Copy the locked device type (Create From on the Add/Edit Device Type screen), then save the copy with a new name.

**The Custom Icon**  :

This icon indicates that a device type or profile is user-created. It may have been created new, created from an existing device type or profile, or created by editing an unlocked custom device type or profile.

### Setting the Screen Resolution

Depending on the screen resolution you use, some of the Profile Editor screens may take up the entire viewing area. We recommend that you use at least 1024 x 768 resolution.

Options are:

| Setting | Resolution (height x width) |
|---|---|
| VGA | 640 x 480 |
| SVGA | 800 x 600 |
| XGA | 1024 x 768 |
| SXGA | 1280 x 1024 |
| WUXGA | 1920 x 1200 |
| User | user-defined height and width |

You can also auto-hide the taskbar to provide more room.


© 2011 Schneider Electric All Rights Reserved

# Main Menu Options

The main menu options (File and Settings) on each of the major tabs of the Profile Editor are described below.

| Field Name/Valid Entries | Comments |
|---|---|
| File > Save | Saves any current changes. |
| File > Create CSV file | Creates .CSV file of basic tag data. Store the file in a folder you designate. Can be viewed in Excel. |
| File > Print Tag Selections | Displays a print preview of all of the tags for this device type. You can then print the spreadsheet. |
| File > Import | You can import projects from other instances of the Profile Editor. These must be PLS or ICD files.<br>To begin importing, click File > Import. For ,more information, see *Importing and Exporting Project Files* on page 69. |
| File > Export | You can export a PLS or ICD file to be used in another instance of the Profile Editor, or to be used as a backup.<br>To begin exporting, click File > Import. For ,more information, see *Importing and Exporting Project Files* on page 69 |
| Settings > Display Advanced Properties | Causes additional “advanced information” columns to display. |
| Settings > Remove Import Templates | You can delete any import template that has been added to the project. To add import templates, see *Using Import Templates* on page 76. |
| Settings > Set Up Custom Tags | Displays the Add/Edit Custom Tags screen. See *Setting Up Custom Tags* on page 42 for a description of this screen. |
| Settings > Set Up Device Type Categories | Displays the Set Up Device Type Categories. See *Setting Up Device Type Categories* on page 39 for a description of this screen. |
| Settings > Set Up Engineering Unit Templates | Displays the Set Up Engineering Unit Templates screen. Click *Setting Up Engineering Templates and Selecting Conversions* on page 28 for a description of this screen. |
| Settings > Set Up Trend Definitions | Displays the Set Up Trend Definitions screen. Click *Setting Up Trend Intervals* on page 57 for more information. |

© 2011 Schneider Electric All Rights Reserved

# Adding Engineering Unit Templates, Units, and Conversions

An engineering unit is a part of a tag. Use engineering unit templates to simplify the conversion between base units and their conversions (such as inches to centimetres) and to provide consistency in recording data in reports and on-screen viewing. For example, in one project you might want to see amperes reported as kiloamps. In another, you might want to see amperes as milliamps. You will use the Units screens to determine the conversion for standard units and custom units (tied to custom tags) that you create.

You can also create templates to organize user-created unit/conversion pairs. Each template will include all of the predefined engineering units and conversions, as well as the ones you assign to it. These templates can then be used in system projects (see the Set Up Project tab for creating projects).

In this section, you can learn about:

*Setting Up Engineering Templates and Selecting Conversions* on page 28

*Adding or Editing a Base Engineering Unit or Conversion* on page 30

## Setting Up Engineering Templates and Selecting Conversions

Use the Set Up Engineering Unit Templates screen to add, edit, or delete an engineering unit template, and to begin adding, editing, or deleting templates, engineering units, and conversions to the system. To view the Set Up Engineering Units screen, click Settings > Set Up Engineering Unit Templates. The following table describes the parts of the Set Up Engineering Unit Templates screen (it assumes that Display 'Advanced' Fields is checked). Detailed instructions are after the table.

| Field Name | Valid Entries | Comments |
|---|---|---|
| Options box | Create New | Click to begin creating a new engineering unit template. |
| | Create From | Click to create an engineering units template that is based on an existing template. |
| | Edit Existing | Only available if you have added a template.<br>Click to edit an engineering unit or its conversion. |
| | Delete | Only available if you have added a template.<br>Click to begin deleting an engineering unit and its conversion. You cannot delete a locked template. |
| Unit Template to Create From | From the drop-down menu, select the template you wish to copy, in order to create a new template. | This field is live only when *Create From* is chosen as the option. The new template will initially include all of the units/conversions of the original; but you can add units and change the conversion settings. |


© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Unit Template Name | This field is blank if you selected *Create New* or *Create From*; type the name of the new template.<br>A name displays if you have selected a template to edit; you can change the name.<br>A name displays, but it is greyed out if you selected a template to delete.<br>Click *Save* to save the changes you make. | When creating a new template or creating from an existing template, type the name of the new template.<br>To change the name of an existing template, choose it from the *Unit Template to Delete* menu, then change the name here. |
| Lock this Template | Click to prevent the template from being edited in the future. | The only way to "edit" a locked template is to delete it, and add back a new one with the edits entered. |
| Display Associated Projects | Live only when in "Edit" mode. Displays all projects that use this template. | You only need this if you want to delete a template that is associated with a project. Note the projects that display in the list, then go to the Set Up Project tab. For each project that you noted, change the unit template. |
| Display 'Advanced' Fields | Check this box to display additional columns of information about the template. | Unchecked: displays the unit and its abbreviation only. Checked: displays also the conversion, and its abbreviation and multiplier. |
| Selected Unit | n/a | Many standard units are pre-defined; they cannot be edited or deleted. To add a unit or edit a user-created unit, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30. |
| Abbreviation | n/a | Added for the unit when the base unit was created. To edit a user-created unit, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30. |
| Selected Unit | Click the down arrow to display and select the preferred conversion for the unit. | Many conversions are pre-defined. To add or edit a conversion unit, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30.<br>Fahrenheit to Celsius temperature conversions must be handled by editing Cicode (Citect.ini). |
| Abbreviation | n/a | This is abbreviation for the selected unit. When the Selected Unit is changed, this field changes accordingly. |
| Multiplier | n/a | Added for the unit and for the conversion when the base unit was created. Pre-defined units/conversions cannot be changed. To edit a user-created unit, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30. |
| Add/Edit Units button | Click to display the *Add/Edit Units* screen. | Use that screen to add units/conversions, or to edit user-created units/conversions. |

### Creating or Editing an Engineering Unit Template

To create or edit a template:

1. From the Define Device Type Tags tab, click Settings > Set Up Engineering Unit Templates.
2. Click Create New or Create From to begin adding a template. Click Edit Existing to edit a template that you have added.
   a. If you are creating a new template, type the name in the Unit Template Name field.
   b. If you are creating from an existing template or editing an existing template, choose that template from the drop-down list. Type the new of the new template in the Units Template Name field.

© 2011 Schneider Electric All Rights Reserved

c. If you are editing an existing template, choose that template from the drop-down list.

3. Click Save to save a new or edited template, or click Save & Exit to save the template and close the screen. See the next section, Choosing Conversions, to change the unit conversions. To add new unit conversions, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30.

### Selecting Conversions

Use this screen to select unit conversions. To add a conversion, see *Adding or Editing a Base Engineering Unit or Conversion* on page 30.

To select a conversion:

1. From the main window of the Profile Editor, click Settings > Set Up Engineering Unit Templates.
2. Click Edit Existing, then select the template for which you want to select unit conversions.
3. Locate the first base unit you want to change. In the Selected Unit column, click the down arrow to select the conversion you want to use. Fahrenheit to Celsius temperature conversions must be handled by editing Cicode (Citect.ini file).
4. Repeat step 3 for all units that you want to change.
5. Click Save to save the change, or click Save & Exit to save changes and close the screen.

### Deleting a Template

You cannot delete either the Standard Template or a locked template.

To delete a template:

1. From the Define Device Type Tags tab, click Settings > Set Up Engineering Unit Templates.
2. Click the Delete radio button, on the left, to delete a template.
3. Choose the template from the drop-down list.
4. Click Delete, on the right, to delete the selected template. At the Confirm Delete prompt. click Yes.

## Adding or Editing a Base Engineering Unit or Conversion

Use the Add/Edit Units screen to add, edit, or delete base units and conversion units for custom tags. You cannot make any changes to predefined units (those that are greyed out). At the Set Up Engineering Units screen, click Add/Edit Units. The following table describes the fields of the Add/Edit Units screen. Detailed instructions are after the table.

| Field Name | Valid Entries | Comments |
|---|---|---|
| Unit Conversions | n/a for pre-defined units/conversions (greyed out)<br>Select user-created units to begin edits. | All base engineering units and their conversions display. Greyed-out items are predefined; they cannot be edited or deleted. Note that predefined units can have custom conversions, which are editable. |
| Add Base | Click to begin adding a new base unit. | The Base Unit and Base Unit Abbreviation fields become live. |


© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Add Conversion | Click to begin adding a conversion to a base unit. | The Base Unit field displays the unit you highlighted; the Conversion Unit, Conversion Abbreviation, and Multiplier fields become live. |
| Edit Unit/ Delete Unit | Click to either edit a custom unit/conversion, or to delete it. | These buttons are only live when you select a custom, unlocked, unit. |
| Base Unit | When editing a unit/conversion, select the unit from this drop-down menu.<br>When adding a new base unit, type the name. | Used in the Profile Editor only; not passed to PowerLogic SCADA projects for graphics viewing. |
| Base Unit Abbreviation | Type the abbreviation for the selected base unit. | If there is no conversion, this is passed to PowerLogic SCADA for viewing graphics. |
| Conversion Unit | Type the name of the conversion unit, e.g.,milliamps, when amps is the base unit. | Used in the Profile Editor only; not passed to the PowerLogic SCADA project for graphics viewing. |
| Conversion Abbreviation | Type the abbreviation for the conversion unit. | This is passed to the PowerLogic SCADA project for viewing graphics. |
| Multiplier | Use this field to determine the number of base units that are in the conversion unit.<br>Type the multiplier "M," where Conversion Unit x M = Base Unit. | Example: There are 1,000 bytes in a kilobyte; so, the conversion unit multiplier is 1000, If you have 17.3 kB,<br>17.3 x 1,000 = 17300 bytes |

### Adding a Base Engineering Unit and Unit Conversion

To add an engineering unit and/or conversion:

1. Click Add Base.
2. In the Base Unit field, type a unique name for the new unit.
3. In the Base Unit Abbreviation field, type an abbreviated name.
4. Click Save.
5. To add a conversion, highlight the name of the Unit, then click Add Conversion.
6. Type the name of the Conversion Unit.
7. Type the name of the Conversion Abbreviation.

### Editing a Base Engineering Unit or Conversion

Changes are global, for all templates. You cannot change any predefined engineering units/conversions (greyed out).

To edit a unit or conversion:

1. To edit the unit: With the base unit highlighted, click Edit Unit. You can edit the base unit and base unit abbreviation. Click OK to save the changes.
2. To edit a conversion: With the conversion highlighted, click Edit Unit. You can edit the conversion unit, abbreviation, and multiplier. Click Save to save the changes or click Save & Exit to save the changes and close the screen.

### Deleting a Base Engineering Unit or Conversion

Deletions are global, for all templates. You cannot delete units any predefined units/conversions (greyed out).

© 2011 Schneider Electric All Rights Reserved

To delete a unit or conversion:

1. To delete a unit: With the base unit highlighted, click Delete Unit. At the Confirm Delete prompt, click Yes. Click OK to close the screen.
2. To delete a conversion: With the conversion highlighted, click Delete Unit. At the Confirm Delete prompt, click Yes. Click Save & Exit to save the changes and close the screen.


© 2011 Schneider Electric All Rights Reserved

# Defining Device Types and Tags

The Define Device Type Tags tab and its related screens are used to define device-related data: custom tags, device types, and base units/conversions. You will use several screens to add and manage this data.

To access these screens, click the Define Device Type Tags tab on the main Profile Editor screen.

In this section, you can learn about:



## Device Type Screens and Workflow

On the Define Device Type Tags tab, follow these general steps to add tags and devices to your system:

1. To manage the units and unit conversions that you will use (such as amperes into milliamperes), see *Adding or Editing a Base Engineering Unit or Conversion* on page 30.
2. To manage the tags you will use, see *Defining Device Types and Tags* on page 33.
3. To add and edit custom tags, see *Adding Custom Tags and Tag Addresses* on page 42.
4. To add or edit device types, see *Adding, Editing, or Deleting a Device Type* on page 35.
5. To establish device type categories and subcategories, used in reporting, see *Printing the .CSV File* on page 39.
6. To edit tag addresses, see *Editing Tag Addresses* on page 46.

The PowerLogic SCADA system uses the IEC61850 tag naming convention. For information about the types of IEC61850 tags and their configuration, see *Understanding Tags* on page 78.

## Using the Define Device Type Tags Tab

The Define Device Type Tags tab displays device types and the tags that may be associated each device type. This includes real-time, onboard alarm, control, and reset tags. Most of the fields on this tab are read only (they can be changed on other screens). The following table describes this tab. The tags listed assume that Advanced Properties has been checked. Not all elements appear on every sub-tab.

| Field Name/Valid Entries | Comments |
|---|---|
| Device Type Name/Select the device type. | Each device type includes a different number of tag categories, which also changes the list of tags that display. |

© 2011 Schneider Electric All Rights Reserved

| Field Name/Valid Entries | Comments |
|---|---|
| | The device list includes the default device types, as well as any that have been created for this system. |
| Add/Edit button/Click to open the Add/Edit Device Type screen. | Provides a means of adding new device types and editing custom device types (user-created device types that are not locked). Also provides a means of adding new custom tags and editing existing tags. |
| Locked/Custom icons:<br>  | Locked icon indicates that the list of selected tags cannot be edited. Custom icon indicates that the device type was created by a user. See *Locked and Custom Icons* on page 26 for complete information. |
| **Tag groups (left-hand pane)** | |
| Select a tag group; the tags included in that group display on the right. | Each tag belongs to a group. The group is determined when the device is added to the system. For custom tags, this is on the Add/Edit Custom Tags screen. Tags for standard device types are pre-determined and cannot be changed.)<br>**Note**: If a tag group displays in red copy, there is at least one address that is not valid for the tag to which it is assigned. To correct this issue, click the tag group, ensure that Display Advanced Properties is selected, then scroll down through the tags in the right-hand column. The tags that have invalid addresses will have the "Edit…" displayed in red. Click this field to open the Edit Address page; correct the errors in the address. |
| **Tag tabs: Real Time, OnBoard Alarm, Control, and Reset** | |
| Click a tab to view the tags of that type that are included for the selected device type.<br>If the device type is not locked, you can use the Add/Edit Device Type screen to edit the list of tags. | |
| Tag Description (all tag types)/Display only | This is the tag name, hard-coded for standard tags. For custom tags: The name is from the Tag Name field in the Add/Edit Custom Tags screen. |
| Units/Display only | Lists the abbreviation, added when creating the engineering unit template. |
| IEC Tag Name/Display only | Tag name that conforms to IEC61850 standard. See *Understanding Tags* on page 78 for more information. |
| Type (Real Time only)/Display only | Displays the data type chosen when the tag was created. |
| Address (not Control tags)/<br>To edit, click the Edit Address link. | Displays the address information for this tag, including elements such as type of register, number of registers, and scaling and bitmasking data. *Understanding Tags* on page 78 for a detailed description of address construction. |
| Normally Closed (Control tags only)/<br>Check the box to invert the functionality of the control. See description. | For a control with one command, writing a 1 to the tag will cause the command to occur. (This option is greyed out.)<br>For a control with two commands that is either static or normally open, writing a 1 to the tag will cause the first command to occur; writing a 0 will cause the second to occur. (Checkbox not checked.)<br>For a control with two commands that is normally closed, writing a 1 to the tag will cause the second command to occur; writing a 0 will cause the first command to occur. (Checkbox checked.) |
| Edit Addr/Click to display Edit Address screen. (Real Time and Onboard Alarm only) | Provides the means of changing the elements of an unlocked real-time tag address (for example, the number of registers, their numbers, and whether they are consecutive).<br>See *Editing Tag Addresses* on page 46. for detailed information. |
| Register 1/Display only (Real Time tags only) | This field contains first register used to store this tag. If there are additional registers, they are indicated in the address. The total number of registers is listed in the Num Registers column. This field allows you to verify and/or change the value of Register 1 without having to open the Edit Address screen. Note: If you enter a number that is not compatible with other address<br>settings, you are prompted to go to the Edit Address screen. |


© 2011 Schneider Electric All Rights Reserved

| Field Name/Valid Entries | Comments |
|---|---|
| Num Registers/Display only (Real Time tags only) | Displays the number of registers used by this tag. |
| Formatting/Select the format type from the drop-down list (Real Time tags only) | After you change formatting for a tag and move the cursor to another field, you are asked whether you want to open the Address Editor. If you click No, the format is unchanged; if you click Yes, the Edit Address screen opens for you to enter the appropriate changes for this tag. See *Editing Tag Addresses* on page 46. |
| Scaling Register/View or enter the register number (Real Time tags only) | This is entered in the Edit Address screen, but it can be edited here. It is the register used to read the value for scaling. Note: If you enter a number that is not compatible with other address settings, you are prompted to go to the Edit Address screen. |
| Functional Address/Display only (Real Time, Onboard Alarm, Control, and Reset tags) | If you have added a functional address for this tag, it displays here. To add or edit this address, use the Edit Functional Address field.<br>Note: Functional addressing is described in *Appendix 4: Glossary* on page 196. |
| Edit Functional Address/Add the code for the address | Typically used for data concentrators, the functional address is a means of entering the individual data points needed to define multiple addresses. Entered as a formula (must be in C#), it will contain the variables the user must enter when the block is instantiated by the Profile Wizard.<br>A simple example:<br>Address =<br>"T:MV;m:" + (startingpoint + 1005).ToString() + ";L:P:22"<br><br>You would then define "startingpoint" when instantiatiing the profile in the Profile Wizard. |
| Tag ID/Display only/Display only | Assigned by the system when the tag was created. If this is a custom tag, it will be a negative number. |
| Category Type (real-time only)<br>Utility Type (real-time only)<br>Statistical Type (real-time only)<br>Quantity (real-time only) | Each of these types is a real-time filter, added when the tag was created. See *Setting Up Custom Tags* on page 42 for more information. |
| Categorization (onboard alarm only)<br>Subcaategorization (onboard alarm only)<br>Alarm Type (onboard alarm only)<br>Alarm Group (onboard alarm only)<br>Alarm Level (onboard alarm only) | Each of these is an onboard alarm folder, added when the tag was created. See *Setting Up Custom Tags* on page 42 for more information. |
| | |

## Adding, Editing, or Deleting a Device Type

Use the Add/Edit Device Type screen to begin adding, editing, or deleting a device type from the system. To view this screen, click Add/Edit, to the right of the Device Type Name field. The following table describes the parts of the Add/Edit Device Type screen. Detailed instructions are after the table.

| Field Name | Valid Entries | Comments |
|---|---|---|
| Create New | | Click to begin adding a new device type that is not based on an existing type. |
| Create From | | Click to make a copy of any existing device type. |
| Edit Existing | | Click to edit an unlocked device type. |

© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Delete Existing | | Click to delete an unlocked device type that is not associated with a profile. |
| Device Type (to Create From/to Edit/ to Delete) | select type | Select the device type that you want to create from, edit, or delete. |
| Copy Addressing | | Active only when you choose Create From. Check this box if you want to copy the addressing of the “from” device. This gives each tag in the new device type the same address string as the matching tag in the “from” device. |
| Device Type Name | Type or select the name:<br>maximum 32 characters,<br>do not use \ / : * ? < > \| | If creating a device type, type the name. If editing an existing device type, the device type that was selected for editing displays here. You can overwrite the name here to change it. |
| Lock this Device Type | Check to lock device, so that it cannot be edited. | Once locked, you cannot edit a device type. If it is a standard, you cannot edit or delete it. If you added the device type, you can delete it, but not edit it. |
| Device Category | Choose the category for this device. | To create categories, see *Printing the .CSV File* on page 39. Categories are used in the Device Creation Wizard. |
| Subcategory | Choose the subcategory for this device, if needed. | As with categories, subcategories are created in the *Printing the .CSV File* screen. |
| Driver | Select the driver for the device type. | Predefined drivers have been created for all PowerLogic compatible devices, though you may need to use these drivers for multiple device types. For example, you would use the CM4000 driver for a CM3000.<br>Use the Generic Power Device driver for third-party devices. The Controllable Device driver is currently not used.. |
| Display Associated Profiles | (Active only in Edit mode)<br>Click to display a list of profiles that are associated with the selected device type. | This list is ant when you want to delete a device type that is associated with a profile. In Edit mode, select the device type you want to delete; then note the profile(s) associated with the device type.<br>Before you can delete the device type, go to the Add/Edit Device Profile screen, locate each profile in the list. You cannot save an empty profile. so you must either select another device type for it or delete the profile. Then you can delete the device type. |
| IEC Tags | n/a | This list includes all tags that have been added to the system, standard tags as well as custom tags that you have added. Tags are listed in their groups (100ms, Onboard Alarm, Power Factors, etc.). |
| Selected Tags | Select tags from Tags; click the right arrow to move them to this box. | You can move single tags or entire tag groups. They must be moved one at a time (cannot shift+click to select).<br>Note: You cannot deselect tags for a device type if that device is associated with a device profile. |
| Add/Edit Custom Tags | Click to begin adding a custom tag. | Live when creating or editing a tag. Opens the Add/Edit Custom Tags screen. See *Adding Custom Tags and Tag Addresses* on page 42 for instructions.<br>If you add a custom tag here, you are prompted to save the device type. After adding the tag, you have the option of adding that tag to the device type. |

## Adding a Device Type

To add a device type:


© 2011 Schneider Electric All Rights Reserved

1. From the Define Device Type Tags tab, click Add/Edit.
2. In the Device Type Options box, click Create New or Create From.

   If you are creating from another device type, choose it from the Device Type to Create From drop-down list.

   If you are creating from another device type, and you want to use the same addressing, check Copy Addressing.
3. Type a Device Type Name for the new device type; maximum 32 characters. Do not use \ / : * ? < > |
4. If you want to lock the device, preventing anyone from editing it, check Lock this Device Type. This action cannot be undone. If you wish to edit a locked device, you must use the Create From feature to add a new one, then delete the locked one.
5. From the Device Category drop-down list, choose the category to which this device belongs. In addition to predefined categories, you can add custom categories. See *Setting Up Device Type Categories* on page 39 for instructions. Categories are used in the Device Creation wizard, and are a means of shortening the list of devices you must view.
6. From the Subcategory drop-down list, choose a subcategory. As with categories, subcategories are added in the Set Up Device Type Categories screen.
7. From the Driver drop-down list, choose the correct drivers for this device type. Drivers have been added for all of the standard, PowerLogic-compatible, devices.
8. The Display Associated Profiles link displays a list of profiles to which a device type has been added. You will use this link only when you want to delete a device type that has been added to a profile.

   A device type that is associated with a profile will not display in the “Delete Existing” mode. So, to view associated devices, you need to switch to “Edit Existing” mode and display the device type you want to delete; then click this link. You will see a list of all profiles to which this device type has been added. Note all of the profiles. Then, do the following:

   Open the Create Device Profiles tab, choose the appropriate device profile, and click Add/Edit.

### Editing a Device Type

You cannot edit any locked device type. When a device type is locked, the locked icon displays on the Define Device Type Tags tab:

If you want an edited version of a locked device type, you must create a new device type from it and then delete the original device type. Certain “standard” device types can be used to create new types, but they cannot be deleted. Examples: Circuit Monitor 4000, Power Meter 800, and Sepam S42.

There are two ways to edit device types:

- From the Define Device Type Tags tab, select a device type; make these changes:

  Edit the functional address (see *Editing Functional Addresses* on page 41).

  Real Time Tags sub-tab: you can edit the address (see *Editing Tag Addresses* on page 46) and choose a different format.

© 2011 Schneider Electric All Rights Reserved

- From the Define Device Type Tags tab, select the device type you want to edit, then click Add/Edit. Follow through the screens to edit additional information:

  In the Device Type Options box, click Edit Existing.

  Click the Device Type to Edit list to display the Select Device box. Select the device type that you want to edit.

  You can change the device type name, category, subcategory, and driver. You can also lock the device.

  Select tags and tag groups and move them into or out of the Selected Tags list.

  If a device type is associated with a device profile, you cannot deselect tags.

  After all of the appropriate changes are made, click Save to save your current settings.

  To create additional custom tags, click Add/Edit Custom Tags; otherwise, click Save & Exit to save your settings and close the window.

  If you add a tag to a group that is already included in a device type, you must then individually add the tag to that device type.

### Deleting a Device Type

You cannot delete any standard device type or any custom device type that is associated with a device profile; those device types do not display in this option.

To view a list of profiles associated with a device type, switch to the Edit Existing view, then select the device type you want to delete. Click List Profiles associated with this Device Type to display all associated profiles. Before you can delete the device type, return to the Add/Edit Device Profile screen for each profile in the list; either select a new device type for the profile or delete the profile entirely.

To delete a device type:

1. From the Define Device Type Tags tab, click Add/Edit.
2. In the Device Type Options box, click Delete.
3. From the drop-down list, select the device type you want to delete (the list includes only unlocked or custom device types; you cannot delete any of the “standard” device types or custom device types that are associated with a profile.
4. Click Delete. At the Confirm prompt, click Yes.

### Assigning Tags to Generic I/O Points

Device types have default tags that have the appropriate formatting and addressing assigned for all the generic I/O points. It may be necessary to redefine a generic I/O point by assigning it to a tag that has a specific meaning.

**Example 1**: The Branch Circuit Monitor 42 has been configured to read 42 current channels. To assign channel 1 to Current A, follow these steps:

1. From the Branch Circuit Monitor 42 device type, choose the “Ch.01 Current tag.”
2. Note the addressing and formatting for the tag.
3. Locate and add the standard tag that you want to assign to this channel. In the example above, you would add “Current A.”
4. Edit the address of the Current A tag to match the address of Channel 1.

**Example 2**: If the Sepam I11 / I12 have been configured to represent circuit breaker position, you may choose to redefine the tag name. Follow these steps:


© 2011 Schneider Electric All Rights Reserved

1. From the Sepam 40 Series device type, choose tags “Input Status I11” / “Input Status I12.”
2. Note the addressing and formatting for each tag.
3. Locate and add the standard tag that you want to assign to these I/’O points. In the example above, you would add “Device Closed.”
4. Edit the address of the Device Closed tag. In order to create the “device closed” functionality, you must combine inputs 11 and 12 into an enumerated status (choose the Enumerated Status logic code for the indicated address for I11 and I12),

## Printing the .CSV File

For each device type, device profile, or project, you can create a .CSV file that includes the following data:

| Type of File | Data Included |
|---|---|
| Device Type | tag descriptions, IEC tag names, type, and address |
| Device Profile | tag descriptions, IEC tag names |
| Project | data profiles and custom tag names included in the project |

Follow these steps to create the CSV file:

1. Display the device type, profile, or project for which you want the file. For example, to create a CSV file for the Sepam 42 Full device profile, select the Create Device Profiles tab and choose Sepam S42 Full from the drop-down list.
2. Click File > Create CSV File.
3. From the Save As window, choose a location for the file and rename it, if you wish. Click Save.
4. The file is created in the location you specified.
5. For viewing and printing, open the file in Microsoft Excel.

## Setting Up Device Type Categories

Use the Set Up Device Type Categories screen to add, edit, and delete categories. These categories are used in the Profile Wizard to logically group the list of profiles that display, and to make them easier to locate.

When you add device types in the Add/Edit Device Type screen, you associate a category and subcategory with each device.

To view the Set Up Device Type Categories screen, click Settings > Set Up Device Type Categories.

The following table describes the parts of this screen. Detailed instructions are after the table.

© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Categories Options box | Create New | Click to begin adding a new device type that is not based on an existing type. |
| | Edit Existing | Click to begin editing the category or subcategory name. |
| | Delete Existing | Click to begin deleting a category. You cannot delete a category that is associated with a device type. |
| | Category Name | If new: Type the name.<br>If editing or deleting, select the name from the drop-down menu.<br>Predefined categories do not display. Currently, there is one predefined category: Schneider Electric. |
| Subcategories Options box. | As with categories, you can create new, edit existing, or delete. | If new: Type the name.<br>If editing or deleting, select the name from the drop-down menu.<br>You cannot delete a subcategory that is associated with a device type.<br>Predefined subcategories do not display. Currently, the predefined subcategories are: Protection Device, Monitoring Device, and PLC. |

### Adding a Category or Subcategory

To add a category/subcategory:

1. Click Create New in the appropriate box (Categories or Subcategories).
2. In the Name field, type the name of the new category or subcategory.
3. Click OK to save the new entry and close the screen.

### Editing a Category/Subcategory Name

To change the name of a category or subcategory:

1. From the appropriate box, click Edit Existing.
2. From the dropdown menu, select the category or subcategory that you want to edit.
3. Type the new name for this category or subcategory.
4. Click Save to make the change, or click Save & Exit to save changes and close the screen.

### Deleting a Category or Subcategory

Predefined categories/subcategories, or those associated with a device type, do not display for deletion. To delete a device type associated with a category/subcategory, change to the “Edit” view, select the category or subcategory, then click the List Device Types... button. Note the device types and go to the Add/Edit Device Types screen. Change the category/subcategory on that page. Then you can return to the Set Up Device Type Categories screen to delete the category/subcategory.

To delete a category or subcategory:

1. From the appropriate box, click Delete Existing.
2. From the dropdown menu, select the category or subcategory that you want to delete.
3. Click Delete.
4. At the Confirm prompt, click Yes.


© 2011 Schneider Electric All Rights Reserved

5. Click Save to save the change, or click Save & Exit to save changes and close the screen.

## Editing Functional Addresses

If you want to add variables to addressing, use this feature. You can re-use these variables by copying/pasting parts of it into other addresses, then making changes to the code for use in other tags. You will be prompted for these variables in the Profile Wizard.

To access the Edit Functional Address screen, click the Edit Functional Address button for a real time tag, onboard alarm tag, control tag, or reset tag. The fields on this screen are used in this way:

1. Tag Name and Original Address: These fields display from the tag you selected; you cannot edit this information.
2. Device Variables: Click New to begin adding new variable properties. The following fields become live:
   a. Name This name must be in format %NNN%, where NNN includes only letters or underscores.
   b. Description: This required field is free-form. It displays in the Profile Wizard and will help you ensure that you have the correct information entered.
   c. Regular Expression: You can use one of the pre-defined expressions, or you can create your own
   d. Test Value: This will become the default in Citect; use it here for testing the new address.
   e. Help: Use this optional field to add more definition to this address. It displays in the Profile Wizard.
3. Code Body: Enter the code in C# to define the action you want to take place.
4. Return: Type the return statement that you want from C# code. It might look like:

   string.Format("SomeString{0}SomeOtherString", someVariable)
5. Result: Click Test in the lower right corner of the screen. If there is a compile error, check your C# code. Otherwise, the result displays. Verify that it is what you wanted.

© 2011 Schneider Electric All Rights Reserved

# Adding Custom Tags and Tag Addresses

PowerLogic SCADA comes with most of the tags that are needed for each device type. However, you can create custom tags to assign to device types and device profiles. You can also edit address attributes for any tag.

NOTE: To avoid potential communication errors, you should use the Profile Editor to create all custom tags that will communicate with equipment.

In this section, you can learn about:

*Setting Up Custom Tags*

*Editing Tag Addresses* on page 46

*Editing Tag Addresses*

## Setting Up Custom Tags

Use the Add/Edit Custom Tags screen to edit the information about a custom tag. To view this screen, click Add/Edit Custom Tags at the bottom of the Add/Edit Device Type screen; or, from the upper left of the Profile Editor screen, click Settings > Set Up Custom Tags. Detailed instructions are after the table.

| Field Name | Valid Entries | Comments |
|---|---|---|
| Custom Tag Options | Create New | Click to begin adding a new tag. |
| | Create From | Click to begin adding a new tag that is based on an existing custom tag. For example, you might want to change metadata for another custom tag. |
| | Edit Existing | Click to edit the attributes of an existing tag. |
| | Delete Existing | Click to delete a tag (tag cannot be associated with a device type). |
| | Tag to Create From<br>Tag to Edit<br>Tag to Delete | From the drop-down menu, select the tag you want to create from, edit or delete. |
| | Delete button | Live only when Delete Existing is selected. Click to delete the tag. You can only delete custom tags not associated with a device type. |
| Display Associated Device Types | Click to display device types that are associated with this tag. | Live only when in Edit mode. Click to list device types that are associated with this custom tag. Note the device types so that you can delete the tag from them (in the Add/Edit Device Type screen) before you delete the tag. |


© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Tag Name | Type the new tag name; or type the changed name for a tag you are editing. | Maximum 32 characters; can include any alpha or numeric character, as well underscore (_) and backslash (\). Must begin with either an alpha character of underscore. |
| Display Name | Type the name that you want to display when selecting the tag and in other displays. | You might use this field for additional information on the Add/Edit Custom Tags screen. For example, you could describe the data that it logs. It does not display anywhere else in the system. |
| Group | Select the group. | Includes all of the real-time groups (100ms, controls, currents, etc.) plus onboard alarms, resets, and controls. |
| Data Type | Select the data type. | These are PowerLogic SCADA tag data types. They affect the logic codes that are available for display in the Edit Address screen. See *Logic Codes* on page 83 for the data type that matches each logic code. |
| Eng. Units | Select the base unit. | These are the base engineering units for tags, from Engineering Unit Setup. |
| Ignore Unit Conversion | Check to cause the system to ignore any conversions that were added for this tag. | Causes reporting to be according to the base unit, rather than the conversion that was chosen for this tag in the template that is being used. |
| Add Eng Unit | Click to open the Add/Edit Units screen, to add a new engineering unit and/or conversion. | Provides a quicker means of adding an engineering unit that had been overlooked. |
| Citect Format | Select the numerical format. | This is used in for display purposes in PowerLogic SCADA graphics pages. It determines where the decimal displays. For example, if you select #.##, the number 8.12579 would be displayed as 8.12. |
| Polling Priority | Low, Normal, or High | Indicates the level of priority PowerLogic SCADA uses when reading data from devices.<br>Note: In the address field, a priority of 1 = High, 2 = Medium, 3 = Low. |
| Alarm On Text | For onboard alarms only: enter the text for when the alarm is On. | This text displays on the Create Device Profiles tab for the onboard alarm tag, when it is selected for the device type in the profile. It also displays in the Alarm Log. |
| Alarm Off Text | For onboard alarms only: enter the text for when the alarm is Off. | |
| Display 'Advanced' filter selections | Check to display additional filter options in the Real Time Filter and Alarm Filter tabs | Displays several additional filter options on the two "Filter" tabs. These options will be useful in the future for reporting purposes. |
| **You can include additional filters for either real time filters or alarm filters. Though not currently used, these filters will provide metadata for later reporting. Standard tags have some of these filters selected.**<br><br>**A typical usage for these filters might be: when creating a custom tag from an already existing standard tag, you can create matching metadata by using the filters that have been built in to the standard tag.** | | |
| **Real Time Filters tab** (list is expanded when "Display 'Advanced' filter selections" is checked) | | |
| Category Type | Select a category for this tag. | This field provides metadata about the tag. It will be used in future reports. |
| Utility Type | Select a utility type. | Metadata for future use in reporting. |
| Statistical Type | Select a statistical type. | Metadata for future use in statistical reporting. |
| Quantity | Select a quantity. | Metadata for future use in statistical reporting. |

© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| **Alarm Filters tab** (list is expanded when "Display 'Advanced' filter selections" is checked) | | |
| Categorization | Select the alarm category | Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting. |
| Alarm Type | Select the alarm type. | Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting. |
| Alarm Group | Select the group. | Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting. |
| Subcategorization | Select a subcategory. | Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting. |
| Alarm Level | Select the severity level of the alarm. | Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting. |

### Adding a Custom Tag

Follow these steps to create a custom tag that is not already used by PowerLogic compatible devices; for example, to create a tag for a third-party device.

To add a tag:

1. Open the Add/Edit Custom Tags screen (from the Add/Edit Device Type screen, click Add/Edit Custom Tags).
2. In the Custom Tag Options box, click Create New *or* Create From.

   If you are creating from custom another tag, choose it from the Tag to Create From drop-down list.
3. Display Associated Device Types: Live only when in Edit mode. Click to list device types that are associated with this custom tag. Note the device types so that you can delete the tag from them (in the Add/Edit Device Type screen) before you delete the tag. See *Deleting a Custom Tag* on page 45 for instructions on using this button.
4. Tag Name: Type a descriptive name for the new tag (see table above for limitations).
5. Display Name; Type the name you want to use when selecting the tag from lists, or for seeing it in displays.
6. Group drop-down menu: Choose the group to which you are adding this tag. IEC tags are organized into these groups. There are multiple groups for real-time tags, and one group each for onboard alarms, controls, and resets.
7. Data Type drop-down menu: Choose the PowerLogic SCADA data type that matches the logic code for this tag. See *Logic Codes* on page 83 for a list of all logic codes with references to the correct data types.
8. Eng. Units drop-down menu: Choose the unit of measurement used for this tag. These are the base units set up in the Units screens.
9. Ignore Unit Conversion: To report on the base unit for this tag, rather than the unit conversion that has been added in the template you are using, check this box. All reporting will use the base unit for the tag.
10. Add Eng. Unit: To add a unit that is not in the list, click this button.
11. Citect Format: Choose the reporting format, to be used in Citect, from ## to #0.#####.
12. Polling Priority drop-down menu: Choose the priority that PowerLogic SCADA will use when reading data from devices: high, normal, or low. In the address field, a priority of 1 = High, 2 = Normal, 3 = Low.


© 2011 Schneider Electric All Rights Reserved

**Real-Time Filters**: (list is expanded when Display 'Advanced' Filter Selections is checked)

13. Category Type drop-down menu: Choose the category or N/A. Category type populates Custom 2.
14. Utility Type drop-down menu: Choose the type of utility or N/A. Utility type populates Custom 4.
15. Statistical Type drop-down menu: Choose the statistical type or N/A. Statistical type populates Custom 5.
16. Quantity drop-down menu: Choose a quantity or N/A. Quantities populates Custom 6.

**Alarm Filters**: (list is expanded when *Display 'Advanced' Filter Selections* is checked)

17. Categorization Type drop-down menu: Choose the type or N/A. This data is currently used for filtering and sorting alarm data in PowerLogic SCADA. Category type populates Custom 3. Metadata will be used in future reporting.
18. Alarm Type drop-down menu: Choose an alarm type or N/A. This data is currently used for filtering and sorting alarm data in PowerLogic SCADA. Alarm type populates Custom 4. Metadata will be used in future reporting.
19. Alarm Group drop-down menu: Choose a group or N/A. This data is currently used for filtering and sorting alarm data in PowerLogic SCADA. Alarm group populates Custom 5. Metadata will be used in future reporting.
20. Subcategorization drop-down menu: Choose a subcategory or N/A. This data is currently used for filtering and sorting alarm data in PowerLogic SCADA. Subcategorization populates Custom 6. Metadata will be used in future reporting.
21. Alarm Level drop-down menu: Choose the severity level of the alarm. This data is currently used for filtering and sorting alarm data in PowerLogic SCADA. Alarm level populates Custom 7. Metadata will be used in future reporting. This default can be overridden when you are editing the device profile (see *Adding, Editing, or Deleting a Project* on page 66)
22. Click Save to save changes, or click Save & Exit to save changes and close the screen.
23. If you are adding the tag from with the Add/Edit Device Type window, the Select Tag screen displays. Click Yes if you want to add this tag to the currently selected device type; click No to not add it.

### Editing a Custom Tag

You can edit any custom tag. To edit a tag:

1. Open the Add/Edit Custom Tags screen (from the Add/Edit Device Type screen, click Add/Edit Custom Tags).
2. In the Custom Tag Options box, click Edit Existing.
3. You can change any of the tag attributes. (This does not change the tag's assignment status; if it is selected for a device type, it does not move back to the IEC Tags list.)
4. Click Save to save changes, or click Save & Exit to save changes and close the screen.

### Deleting a Custom Tag

You can delete any custom tag that is not associated with a device type.

If the tag is associated with a device type, you must first deselect the tag:

© 2011 Schneider Electric All Rights Reserved

1. Change the option to Edit Existing and display the tag you want to delete.
2. Click Display Associated Device Types to display all device types that include this tag. Make a note of the device types.
3. Return to the Add/Edit Device Type screen. For each device type listed, deselect the tag that you want to delete.

Continue deleting the tag:

1. Open the Add/Edit Custom Tags screen.
2. In the Custom Tag Options box, click Delete Existing.
3. From the drop-down menu, choose the tag you want to delete.
4. Click Delete. At the Confirm prompt, click Yes.
5. Click Save to save the change, or click Save & Exit to save changes and close the screen.

## Editing Tag Addresses

Use the Edit Address screen to edit the attributes of a single tag address. If a device type is locked, you cannot edit any of its tag addresses; they will be greyed out. A thorough discussion of IEC61850 tags and their construction is included in *Understanding Tags* on page 78 and *Logic Codes* on page 83.

NOTE: Case and order are critical in the tag address. Be careful to observe the exact address order. For address order, see *Logic Codes* on page 83. Also, be sure you use the correct case. For example, use **M** for register numbers in hexadecimal, and use **m** for register numbers in decimal.

To view the Edit Address screen, open the Define Device Type Tags tab, choose the device type, then click the Edit... field for the tag that you want to change. The Edit Address screen is different for real-time and alarm tags. Each type of tag (real-time, onboard alarm, reset, and control) is described separately in the following tables.

### Real-Time Tag Addresses

The following table describes the fields of the Edit Address screen for real-time tags. Detailed instructions are after the table.

| Field Name | Entry | Comments |
|---|---|---|
| Data Type | For display only | You can edit this field in the Add/Edit Custom Tags screen. |
| Priority | High, Normal, or Low<br>Logic Code: | You can edit this field either here or in the Add/Edit Tag screen. |
| Logic Code | Select the logic code for this tag. | The logic code list depends on the Data Type for this tag. For more information about logic codes, see *Logic Codes* on page 83. |
| Display Registers in: | hexadecimal/decimal | Click the radio button for the way you want to view register information. |
| Module | Select module | Choose the type of module in which the tag is used. Used for Micrologic at this time. |
| Register Type | Select register type | Select the type of register that is to be written or read. |
| Number of Registers | Select the total number of registers for this address (1-10).<br>Is Consecutive, check if the registers are to be consecutive (determined in the logic code). | Enables for editing the appropriate registers in the lines below. |


© 2011 Schneider Electric All Rights Reserved

| Field Name | Entry | Comments |
|---|---|---|
| Fixed Scale/Register Scale | Click the radio button for the correct type of scale. | A fixed scale is the actual value. of the scale. A register scale is the register address where the scale is held.<br>The value will be scaled in this manner: Value x 10x<br>where X = the scale.<br>Scales can only be –10 to 10. |
| Conversion Factor | Enter the multiplier to convert the base units to the desired conversion. | Conversion factors are used for straight multiplication with the value. The conversion factor could also be changed in the Add/Edit Units screen (Settings > Select Units > Add/Edit Units.<br>Conversion factors take this form:<br>####E##. For example,<br>123E-2 becomes 123x10-2<br>which becomes 1.23. |
| Offset | y = ,x + b | y = the final value reported by PLSCADA<br>b = the offset<br>m = the conversion factor<br>x = the original value in the meter<br>b = rarely used, mainly in temperature conversion<br>The offset is added to the final value (after the conversion factor is applied). |
| Register 1-4 | Enter the register number. | Be aware of whether you chose hexadecimal or decimal. Use the same format here. |
| Bitmask for Register 1-4 | For digital input/output tags: Set the bits to 1 or 0 to match the pattern for “True” in the device register. | When all bits match exactly the pattern in the register, the status is True. When any one bit does not match the pattern in the register, the status is False.<br>Note: On PM8s and CM4s, there is a device-specific format, DIgIn and DigOut. In each case, you must first specify the indicator register (which becomes the first register). The second register will have the mask. |
| Invert Result | Check this box to invert. | Will turn False to True or vice versa; typically used for Normally Open or Normally Closed. |

### Onboard Alarm Tag Addresses

The following table describes the fields of the *Edit Address* screen for onboard alarm tags. Detailed instructions are after the table

| Field Name | Entry | Comments |
|---|---|---|
| Tag Name | For display only | This is the tag name, which cannot be changed. |
| File Number | Select the number. | This is the file number for the alarm file on the device.(Sepam has no file number; enter 0.) |
| Module | Select the module. | Choose the type of module in which the tag is used. Used for Micrologic at this time. |
| Unique ID | Choose the identifier. | This unique identifier must be used to ensure that alarms will annunciate correctly. For CM4, PM8, and Micrologic, the unique ID must be decimal. For Sepam, the unique ID is the coil bit address that indicates the alarm; it must be in hexadecimal. |
| Hexadecimal | check box | Check this box if you want to display the ID in hexadecimal, rather than decimal. |
| Has Unique Sub ID | check box | Check if this tag has a unique sub-identifier (Micrologic, CM4000, and PM800 devices). |
| Unique Sub ID | Enter the Sub ID. | Enter the unique sub-identifier. Active only if Unique Sub ID box is checked. |

### Reset Tag Addresses

NOTE: Once the tag is set up, writing a 1 to the tag will cause the “write” to occur.

© 2011 Schneider Electric All Rights Reserved

Standard device types include some pre-defined resets. These pre-defined commands cause proprietary functions within the device. Do not edit these commands.

To add a custom reset that will operate by writing to a register, do the following:

1. From the Add/Edit Custom Tags screen, set the Group to Resets and the Data Type as Digital. Save the tag.
2. Add the new tag(s) to the appropriate device type(s).
3. From the Define Device Type Tags tab, locate the tag and click Edit.

The following table describes the fields of the Edit Address screen for reset tags. Detailed instructions are after the table.

| Box Name | Field Name | Comments |
|---|---|---|
| Tag Information | Command Type | The Command Type and Command to Edit are already selected. |
| | Command to Edit | |
| Data Information box | Data Type: for display only | You can edit this field in the Add/Edit Custom Tags screen. |
| | Priority: High (default) | Cannot be edited. |
| | Logic Code: Select the logic code for this tag. | Choose the appropriate logic code for this tag. See *Logic Codes on page 83*. |
| Device Information box | Display Registers in: hexadecimal/decimal | Click the radio button for the way you want to view register information. |
| | Module | Choose the type of module in which the tag is used. Used for Micrologic at this time. |
| | Register Type | Select the type of register that is to be written or read. |
| Number of Registers | There is only one register for this address. | Enables for editing the appropriate registers in the lines below. |
| Fixed Scale/Register Scale | n/a | Not used for digital logic codes. |
| Conversion Factor | n/a | Not used for digital logic codes. |
| Register 1 | Enter the register number. | Be aware of whether you chose hexadecimal or decimal. Use the same format here. |
| Bitmask for Register 1 | For digital input/output tags: Set the bits to 1 or 0 to match the pattern for “True” in the device register. | When all bits match exactly the pattern in the register, the status is True. When any one bit does not match the pattern in the register, the status is False.<br>Note: On PM8s and CM4s, there is a device-specific forma, DIgIn and DigOut. In each case, you must first specific the indicator register (which becomes the first register). The second register will have the mask. |
| Invert Result | n/a | Not used for resets. |

### Control Tag Addresses

NOTE: For a control with one command, once the tag is set up, writing a 1 to the tag will cause the “write” to occur. For a control with two commands that is either static or normally open, writing a 1 to the tag will cause the first command (ON) to occur; writing a 0 will cause the second (OFF) to occur. For a control with two commands that is normally closed, writing a 1 to the tag will cause the second command (OFF) to occur; writing a 0 will cause the first command (ON) to occur.

Standard device types include some pre-defined controls. For example, Operate (ENERGIZE). These pre-defined commands cause proprietary functions within the device. Do not edit these commands.

To add a custom control that will operate by writing to a register, do the following:


© 2011 Schneider Electric All Rights Reserved

1. From the Add/Edit Custom Tags screen, set the Group to Controls and the Data Type as Digital. Save the tag.
2. Add the new tag(s) to the appropriate device type(s).
3. From the Define Device Type Tags tab, locate the tag and click Edit.

The following table describes the fields of the Edit Address screen for control tags.

| Box Name | Field Name | Comments |
|---|---|---|
| Tag Information | Command Type | For commands that have an opposite (such as On and Off), choose Normally Open/Normally Closed or Static with Off Command. For commands with only one action, choose Static without Off Command. |
| | Command to Edit | If you are editing a command with two parts, use the Command to Edit drop-down menu to select the On Command. |
| Data Information box | Data Type: for display only | You can edit this field in the Add/Edit Custom Tags screen. |
| | Logic Code: Select the logic code for this tag. | Choose the appropriate logic code for this tag. See *Logic Codes on page 83*. |
| Device Information box | Display Registers in: hexadecimal/decimal | Click the radio button for the way you want to view register information. |
| | Module | Choose the type of module in which the tag is used. Used for Micrologic at this time. |
| | Register Type | Select the type of register that is to be written or read. |
| Number of Registers (1) | n/a | Enables for editing the appropriate registers in the lines below. |
| Fixed Scale/Register Scale | Click the radio button for the correct type of scale. | A fixed scale is the actual value. of the scale. A register scale is the register address where the scale is held.<br>The value will be scaled in this manner: Value x 10x<br>where X = the scale.<br>Scales can only be –10 to 10. |
| Conversion Factor | n/a | Not used for digital controls. |
| Register 1 | Enter the register number. | Be aware of whether you chose hexadecimal or decimal. Use the same format here. |
| Bitmask for Register 1 | For digital input/output tags: Set the bits to 1 or 0 to match the pattern for “True” in the device register. | When all bits match exactly the pattern in the register, the status is True. When any one bit does not match the pattern in the register, the status is False.<br>**Note**: On PM8s and CM4s, there is a device-specific forma, DIgIn and DigOut. In each case, you must first specific the indicator register (which becomes the first register). The second register will have the mask. |
| Invert Result | n/a | Not used for digital controls. |

### Editing Address Information

To edit address information for a real-time tag:

1. From the Define Device Type Tags tab, choose a device type (cannot be locked). From the Real Time Tags sub-tab, highlight the tag whose address you want to edit.
2. In the Edit Address column, click “Edit” for the address you want to edit.

   The Edit Address screen displays.
3. You can change any of the tag address attributes. See the preceding table for descriptions of each field.
4. Click *OK* to save changes and close the screen.

© 2011 Schneider Electric All Rights Reserved

### Adding a New Tag Address

You can also add a tag address, when none exists. As with editing addresses, click the Edit Address column for a tag; then follow instructions in the table above.

## Editing Tag Addresses, Generic

This screen displays when you click Edit for an address of a non-PowerLogic compatible device type, such IEC 61850 or DNP3.

The variable tag properties used in this screen are described in a topic in the Citect help file. See Variable Tag Properties for detailed information.


© 2011 Schneider Electric All Rights Reserved

# Creating Device Profiles

Use the Create Device Profiles screens to view and edit profiles for individual devices. Profiles are predefined for the standard devices; you will mostly use this feature to add third-party device profiles. Before you create profiles, you need to be sure that all of the tags and device types that you need are created (see *Defining Device Types and Tags* on page 33). Also make sure that you have added any new units/conversions and device type categories/subcategories that are needed.

In this section, you can learn about:



## Enabling Waveforms

On the Create Device Profiles tab, in the Onboard Alarm Tags sub-tab, there is a Waveform checkbox. Check the box for each alarm tag for which you want to be able to view waveforms. On the device, the alarm must also be set up for the waveform to be captured on event and stored in one of the device's data logs.

To acquire waveforms for Sepam, use the SFT manual. For PowerLogic devices, reference the PMCU help file.

As device information is polled and received by PowerLogic SCADA, the waveform becomes available for viewing. See *Changing the Display Name in an Alarm or Event Log* on page 146 for information on viewing waveforms in the runtime environment.

## Viewing Device Profiles

The Create Device Profiles tab displays all of the tags that are included in each device type profile. It is the starting point for creating/editing device profiles for individual devices. Most of the data on this screen displays for information only; however, to enable waveforms, you need to check the Waveform box (see *Enabling Waveforms* on page 51 for more information). To begin making changes, click Add/Edit. The following table describes the fields on this tab. The tags listed assume that Advanced Properties has been checked. Not all elements appear on every sub-tab. Detailed instructions are after the table.

| Field Name | Valid Entry | Comments |
|---|---|---|
| Tag Groups (left-hand pane) | Click a group to display the groups of tags that have been selected for the chosen device profile. | To associate tags and tag groups with a device type (thus creating a device profile), click Add/Edit. |

© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entry | Comments |
|---|---|---|
| Device Profile | Choose the device for which you want to view profile details. | Device Profiles are created on the Add/Edit Device Profile screen (click Add/Edit). |
| Add/Edit button | Click to display the Add/Edit Device Profile screen. | Use that screen to add device profiles and to associate PC-based alarms and trends. |
| Tag type sub-tabs | Click to display the selected tags for each type of tag: real-time, trend, PC-based alarm, onboard alarm, control, or reset. | Organized according to tag groups. |
| Tag Description | n/a | This is the tag name used when adding the tag. |
| IEC Tag Name | n/a | This is the IEC61850-compatible name created when the tag was added. |
| Waveform (Onboard Alarm) | Check this box as part of the process of enabling waveform viewing in PowerLogic SCADA. | You must also set up the alarm and waveform capture in the onboard files of the device. Waveforms will then be viewable in the runtime environment. |
| Category Type (Real Time) | n/a | These are real-time filters. They provide metadata to be used in future reporting. |
| Utility Type (Real Time) | n/a | |
| Statistical Type (Real Time) | n/a | |
| Quantity (Real Time) | n/a | |
| Categorization (PC Based and Onboard Alarm) | | These are alarm filters. They can be used for filtering and sorting alarm data in the runtime environment. They also provide metadata to be used in future reporting. |
| Subcategorization (PC Based and Onboard Alarm) | | |
| Alarm Type (PC Based and Onboard Alarm) | | |
| Alarm Group (PC Based and Onboard Alarm) | | |
| Alarm Level (PC Based and Onboard Alarm) | From the drop-down list, you can edit the alarm level. | If PC-based and/or onboard alarms are set for this profile, you can change their levels here. |

To view profile information, select the device profile from the drop-down menu. Use the tag tabs (real-time, trend, etc.) to view the tag groups included in this device profile.

To begin adding, editing, or deleting a profile, click Add/Edit.

## Adding, Editing, or Deleting a Device Profile

Use the Add/Edit Device Profile screen to add device profiles to the system. To view this screen, go to the Create Device Profiles tab.

### Adding a Profile

Click Add/Edit. At the Add/Edit Device Profiles screen, check either Create New or Create From. Click Next. See the tables below for detailed help.


© 2011 Schneider Electric All Rights Reserved

## Editing a Profile

You can only edit unlocked profiles. Select the profile to be edited. Information about the profile displays. If an address is incomplete for a device type in the profile, you will see Device Type is incomplete beneath the Add/Edit button. Before you can proceed, you need to complete all addressing for that device.

There are two ways to edit tags:

- From this first screen, you can select a profile and then:
  - Trend Tags sub-tab: choose trend intervals (to create or edit intervals, see *Setting Up Trend Intervals* on page 57).
  - PC Based Alarms or Onboard Alarms sub-tabs: change alarm levels (this will override the default that is set in *Setting Up Custom Tags* on page 42).
  - Onboard Alarms sub-tab: enable waveform capture for on-board alarms (see *Enabling Waveforms for Onboard Alarms* on page 126 for complete instructions on enabling these waveform captures).
  - Onboard Alarms sub-tab: add Alarm On and Alarm Off text. What you enter here will override the default setting that comes from the custom tag (see *Setting Up Custom Tags* on page 42 for more information).
  - Parameters sub-tab: Edit parameters for IEC 61850 driver parameters (see Editing Driver Parameters for more information).

- Click Add/Edit to progress through several screens to edit all aspects of the profile. See the tables below for detailed instructions.

## Deleting a Profile

You can only delete unlocked profiles. Select the profile to be deleted, and click Add/Edit. The greyed-out Add/Edit Device Profile screen displays. Click Delete Existing. Click Delete, then click Yes to the Confirm prompt.

| Field Name | Valid Entries | Comments |
|---|---|---|
| **Profile Options** | | |
| Device Profile Name | Type or select the profile name: maximum 32 characters<br>do not use \ / : * ? < > \| | |
| Description | optional | This adds descriptive information for “tool tip” display. When you hover over the name of the profile on the Create Device Profiles tab, the description displays. It also displays with the profile name on the Selected Device Profiles sub-tab on the Set Up Project tab.<br>Click Next to be able to select devices. |
| Available Devices | Click the left and right arrows to add devices or remove devices from the profile. | You can add entire device groups (composite, monitoring, protection) or individual devices form each group. |
| **Click Next to select/edit tags for this profile:** | | |

© 2011 Schneider Electric All Rights Reserved

| Field Name | Valid Entries | Comments |
|---|---|---|
| Device Type Tags | For each device, choose the tags you want to include in this profile. | Choose entire categories (Controls, Currents, Resets, etc.), or individual tags from each category. |
| Override Tag Name | Select the tag name that you want to use as an override. | Highlight the tag, then click Override Tag Name.<br>From the Select Tag window, choose the tag you want. If necessary, enter a search term, then click Search to display related tags.<br>Choose the tag, then click OK. |
| Is Device Tag | Check to read data from the physical device, not the composite device. | Composite device types only: Example: if the same tag is in three devices, and you set PC-based alarms for each device, you need to be able to determine which device has a problem in runtime. To prevent confusion, check Is Device Tag to cause PowerLogic SCADA to report the tag for its physical device, rather than the composite device. |
| **Click Next to select PC-based and Trend assignments.** | | |

| Field Name | Valid Entries | Comments |
|---|---|---|
| PC-Based | Check to enable PC-based alarming for the highlighted tag | When the project is added to the PowerLogic SCADA project, PC based alarms are added to the Analog Alarms or Digital Alarms file. |
| Trend | Check to enable PC-based alarming for the highlighted tag | When the project is added to the PowerLogic SCADA project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the PowerLogic SCADA project.<br>There are two different intervals for scanning trend tags. All selected tags are scanned every 15 minutes with FIFO storage of 12 months. For the following tags, there is an additional “short” scanning interval of 5-seconds, with FIFO storage of two weeks:<br>Current A, Current B, Current C, Voltage A-B, Voltage B-C, Voltage C-A, Power Factor Total, Apparent Power Total, Reactive Power Total, Real Power Total, and Frequency.<br>For instructions on changing the “short” scan interval settings, see *Trend Tag Scan Intervals* on page 134 |
| Driver Parameters | Check the appropriate parameters. | For some IEC 61850 device types, this column is populated with parameter options. Select the approrpriate ones for this profile. The parameters allow you to edit datasets and report control blocks. To edit existing datasets and report control blocks, go to the Create Device Profiles tab. Click the Prameters sub-tab. See …. for more information. see *Editing IEC 61850 Datasets* on page 57and *Editing IEC 61850 Report Control Blocks* on page 58. |
| Close Wizard | Check to return to the first page of the Create Device Profiles tab. | When you check this box and then check Finish, you return to the Create Device Profiles tab. When this box is not checked, click Finish to the Add/Edit Device Profile screen. |
| **Click Finish to complete the profile.** | | |

## Adding a Device Profile

To add a profile:

1. Open the Create Device Profiles tab (from the Create Device Profiles tab, click *Add/Edit*).
2. In the Profile Options box, click Create New or Create From.

   If you are creating from another device profile, choose it from the Device Profile to Create From drop-down menu.
3. Click Next to make the name and description fields live.


© 2011 Schneider Electric All Rights Reserved

4. Type a unique Device Profile Name, maximum 32 characters; do not use \ / : * ? < > |
5. If you want to lock this profile, preventing anyone from editing it, check the Lock this Device Profile box. This action cannot be undone. If you wish to edit a locked profile, you must use the Create From option to add a new one, then delete the locked one.
6. (optional) Type a device description. This will display as a tool tip in later screens.
7. Click Next again to make the remaining fields live.
8. From the Available Devices list, highlight the first device or device group (Protection, Monitoring, Composite) to be included in this profile. Click the right arrow button to move it to the Selected Devices box. You must select and move devices or device groups one at a time (no shift+click to select multiples).
9. If you will want to import this project into another instance of the Profile Editor, see Adding Project Parameters.
10. When you have all of the devices you want, click Next.
11. From the Device Type Tags list on the left, select the tags you want to include in this profile. You can select entire tag groups or individual tags from a group; but you must select them one at a time.
12. After each addition, the tag or tag group displays in the Selected tags box. You can override any tag name (typically for generic I/O devices with multiple tags, such as inputs, for which names alone would not be intuitive in runtime. To override a tag, select it, then click Override Tag Name. Choose the tag you want. Click OK. The new tag will correctly display the value of the original tag, but will take the appearance of the override tag (description, metadata, etc.).
13. The final column, Is Device Tag, displays only for composite devices. Check this box to tie a tag back to its actual physical device. For example, if the same tag is in three devices, and you set PC-based alarms for each device, you need to be able to determine which device has a problem in runtime. To prevent confusion, check Is Device Tag to cause PowerLogic SCADA to report the tag for its physical device, rather than the composite device.
14. When you have selected all tags, click Next.

    NOTE: If you have duplicate tags from multiple devices, you need to resolve this by using an override for one of the tags.
15. On the next page, choose whether each tag will have a PC-based alarm and/or trend associated with it.

    When the project is added to the PowerLogic SCADA project, PC based alarms are added to the Analog Alarms or Digital Alarms file.

    When the project is added to the PowerLogic SCADA project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the PowerLogic SCADA project.

    By default, there are two different intervals for scanning trend tags. All selected tags are scanned every 15 minutes with FIFO storage of 12 months. For the following tags, there is an additional “short” scanning interval of 5-seconds, with FIFO storage of two weeks:

    Current A, Current B, Current C, Voltage A-B, Voltage B-C, Voltage C-A, Power Factor Total, Apparent Power Total, Reactive Power Total, Real Power Total, and Frequency.

    For instructions on changing the “short” scan interval settings, see *Trend Tag Scan Intervals* on page 134.

© 2011 Schneider Electric All Rights Reserved

To change a trend interval for a tag, see *Selecting Trend Intervals* on page 56To add additional trend tags, see *Setting Up Trend Intervals* on page 57.

16. The Driver Parameters box contains options that you can check for IEC61850 devices. If a device includes datasets and report control blocks, you can edit the information on the *Editing IEC 61850 Datasets* on page 57 and *Editing IEC 61850 Report Control Blocks* on page 58 screens.
17. Check the Close Wizard box, and click Finish to return to Create Device Profiles tab/ Or, leave it unchecked, and click Finish to return to the Add/Edit Device Profile screen.

### Editing a Device Profile

Only unlocked profiles are available for editing.

To edit a profile:

1. Open the Create Device Profiles tab (from the Create Device Profiles tab, click Add/Edit).
2. In the Profile Options box, click Edit Existing.
3. From the drop-down menu, choose the profile you want to edit.
4. You can change any of the attributes that have been selected for this profile.
5. Click Save to save the change, or click Save & Exit to save changes and close the screen.

### Deleting a Device Profile

You cannot delete standard profiles or custom profiles that have been associated with projects. To delete a custom profile that is associated with a project, you need to go to the Set Up Project tab.

To delete a profile:

1. Open the Create Device Profiles tab (from the Create Device Profiles tab, click Add/Edit).
2. In the Profile Options box, click the Delete Existing radio button.
3. From the drop-down menu, highlight the profile you want to delete.
4. Click Delete.
5. At the Confirm prompt, click Yes.
6. Click Save to delete the profile, or click Save & Exit to delete the profile and close the screen.

## Selecting Trend Intervals

Use the Select Trend Intervals screen to edit settings for existing trends for specific device profile/tag combinations. To create new trends, see *Setting Up Trend Intervals* on page 57.

To change a trend interval, follow these steps:

1. On the Create Device Profiles tab, choose the device profile, then click the Trend Tags sub-tab.
2. Locate the tag for which you want to change the trend. Click Edit.
3. At the Select Trend Intervals screen, you can select one or all of the interval options.
4. Click OK.


© 2011 Schneider Electric All Rights Reserved

## Setting Up Trend Intervals

For any of the trend definitions that are in the system, you can add or edit trend intervals.

### To add a trend interval:

1. Access the screen: Click Settings > Set Up Trend Definitions.
2. From the Set Up Trend Definitions screen:
   a. Click New to begin adding a new trend
   b. Select a trend, then click Copy to create a new trend from an existing trend.
3. Type a name: must begin with either an alpha character (A-Z or a-z) or the underscore character (_). Any following characters must be either alpha characters (A-Z or a-z), digit characters (0 - 9), backslash characters (\), or underscore characters (_).
4. Type the appropriate information in the following fields. See Trend Tag Properties in the CitectScada.chm help file for details on each field.
5. Click OK to save the new trend definition, which displays in the left-hand box.

### To edit a trend interval:

1. From the Set Up Trend Definitions screen, select the trend name, then click Edit.
2. You can edit any of the fields except the trend name.

### To delete a trend interval:

1. From the Set Up Trend Definitions screen, highlight the name of the trend to be deleted.
2. Click Delete, then click Yes when you are asked to confirm.

## Editing IEC 61850 Datasets

Use this screen to add and edit IEC 61850 tag datasets to a profile. To access this screen, display the Create Device Profiles tab for a device that includes ICD files. Click the Parameters sub-tab, then click Edit on the DataSets line.

NOTE: Not all ICD files allow you to add, edit, or delete datasets. If all fields are greyed out, you will not be able to change the set.

In the upper left corner are the device profile name and device type names that come from an imported ICD file. All of the entry fields are initially greyed out. The device type datasets (upper box) are resident in the ICD. The device profile datasets (lower box) have been created or copied from other datasets in the device type or device profile.

### Creating a New DataSet

1. To begin creating a dataset, click Create New beside the Device Profile DataSets box.

   The fields on the right side of the screen become live.
2. Type a name and description for the new dataset. These are free-form fields, but they must comply with IEC 61850 standards.

© 2011 Schneider Electric All Rights Reserved

3. Choose the appropriate logical device, then choose the logical node for that device.
4. Choose the functional constraint for the content. This will filter the display of device type objects/topics in the box below.

   When you choose All, you must then choose an object that already has a functional constraint in it. If you choose a specific constraint, the list of available objects is filtered to display only those that include that constraint.
5. From the Device Type Objects, choose the appropriate objects for this profile.
6. Click OK.

   The new dataset is added in the lower left, to the Device Profile list.

### Creating a DataSet from an Existing DataSet

You can create a new dataset either from one that resides in the ICD (from the device type) or from the device profile.

1. To begin creating a dataset from another block, click the dataset (either device type or device profile) to be used as the starting point for the new dataset. Click Create From.
2. Make the appropriate changes. You must change the name. All datasets in a single profile must have unique names.
3. Click OK.

   The new name displays under the Device Profile List.

### Copying a DataSet to a Device Type

This feature will not typically be used. If, however, you delete a dataset from the device type, but later decide you want to add it back, follow this procedure. (You cannot delete datasets that are used by a report control block.)

1. From the Device Type DataSets box, highlight the dataset you want to add back.
2. Click Copy To.

   The dataset displays under the Device Type list in the Device Profile DataSets.

### Editing and Deleting DataSets

You cannot edit or delete datasets that are being used by a report control block or those that belong to the device type.

To edit a dataset, highlight its name, then click Edit. Make the desired changes, then click OK.

To delete a dataset, highlight its name. Click Delete, then click OK.

## Editing IEC 61850 Report Control Blocks

Use this screen to edit report control blocks for device type information that comes from imported ICD files. To access this screen, display the Create Device Profiles tab for a device that includes ICD files. Click the Parameters sub-tab, then click Edit on the Report Control Blocks line.


© 2011 Schneider Electric All Rights Reserved

NOTE: Not all ICD files allow you to add, edit, or delete report control blocks. If all fields are greyed out, you will not be able to change the set.

In the upper left corner are the device profile name and device type names that come from an imported ICD file. All of the entry fields are initially greyed out. The device type report control blocks (upper box) are resident in the imported ICD file. The device profile report control blocks (lower box) have been created or are copied from report control blocks in the device type or device profile.

## Creating a New Report Control Block

1. To begin creating a new report control block, click Create New beside the Device Profile Report Control Blocks box.

   The fields on the right side of the screen become live.
2. Type a name and description for the new report control block, conforming to the IEC 61850 naming conventions.
3. Choose the appropriate dataset for this block. Datasets are added/edited in the Add/Edit DataSets screen, accessed from the Parameters sub-tab on the Create Device Profiles tab.
4. Type a report ID, again conforming to the IEC 61850 convention.
5. ConfRev determines the version number of the report control block.
6. If this is a buffered block (BRCB), check Buffered and enter the time and integrity period. (Indexing is currently unavailable in PowerLogic SCADA).
7. Check the appropriate boxes for trigger conditions and report content.
8. Click OK.

   The new report control block is added in the lower left, to the Device Profile list.

## Creating a Report Control Block from an Existing Report Control Block

You can create a new report control block either from a block that resides in the ICD (from the device type) or from the device profile.

1. To begin creating a block from another block, click the report control block (either device type or device profile) to be used as the starting point for the new block. Click Create From.
2. Make the appropriate changes. You must change the name. All report control blocks in a single profile must have unique names.
3. Click OK.

   The new name displays under the Device Profile List.

## Copying a Report Control Block to a Device Type

This feature will not typically be used. If, however, you delete a report control block from the device type, but later decide you want to add it back, follow this procedure.

1. From the Device Type Report Control Blocks box, highlight the block you want to add back.
2. Click Copy To.

© 2011 Schneider Electric All Rights Reserved

The report control block displays under the Device Type list in the Device Profile Report Control Blocks.

### Editing and Deleting Report Control Blocks

You cannot edit or delete datasets that belong to the device type.

To edit a report control block, highlight its name, then click Edit. Make the desired changes, then click OK.

To delete a report control block, highlight its name. Click Delete, then click OK.

## Creating a Composite Device Type

A composite device is a device profile that includes more than one device type. Each device type can use its own protocol for communication.

With the composite device type, the user can use two devices for a single monitoring point. For example, a circuit breaker and a monitoring device can provide data to this single point. Because PowerLogic SCADA combines the functionality of the multiple devices, end users only need to consider a single device when analysing a location in their system.

Follow these steps to create the composite device type:

1. From the Create Device Profiles tab, click Add/Edit.
2. At the Add/Edit Device Profile screen, choose whether you are creating a new device or creating from an existing device. If you are creating from a device type, select it. Click Next.
3. Still on the Ad/Edit Device Profile screen, give the composite device type a name. Optionally, add a description (which will become a tool tip display in later screens). Click Next.
4. Choose the device types to be in the composite. Click Next.

   The Add/Edit Device Profile displays with only device type tags available for selection.
5. Add the tags you need for each device type listed on the left. To add all of the tags for a device type, highlight the device type name and click the right green arrow.

   You may find, especially when dealing with generic I/O, that the tag name is not descriptive enough to determine what it is when reading data in runtime mode. Thus, you may want to override the generic name with something more meaningful.

   For example, .a device may have ten inputs: Ind1, Ind2, Ind3, etc. Using those names, you have no idea what each input is reading. If you override the tag, the tag's value will still come from the original tag (it still keeps the addressing from the device); however the tag's appearance (name, metadata, display name, etc.) will be taken from the new tag.
6. To override a tag:

   a. Highlight the tag, then click Override Tag Name.
   b. From the Select Tag window, choose the tag you want. If necessary, enter a search term, then click Search to display related tags.
   c. Choose the tag, then click OK.


© 2011 Schneider Electric All Rights Reserved

Only for composite devices, the *Is Device Tag* check box displays. Use this box to tie a tag back to its actual physical device. For example, you might have the same tag in each of three devices, and you want to set PC-based alarms for each one. Normally, the composite device would generate a single alarm, but you would not be able to specify which physical device has the problem. To prevent confusion, you would check the *Is Device Tag*, which will cause PowerLogic SCADA to report this tag for its physical device.

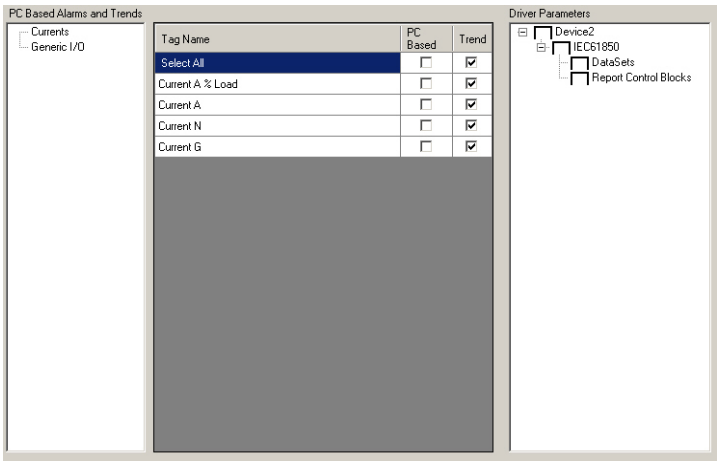
7. Check *Is Device Tag* to read this tag as specific to the physical device, not the entire profile..
8. Click Next to begin selecting tags for PC-based alarms and trends.
9. For each tag in the profile, determine whether it should have a PC-based alarm and/or trend associated with it. Check the boxes as appropriate.

   When the project is added to the PowerLogic SCADA project, PC based alarms are added to the Analog Alarms or Digital Alarms file.

   When the project is added to the PowerLogic SCADA project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the PowerLogic SCADA project.

   There are two different intervals for scanning trend tags. All selected tags are scanned every 15 minutes with FIFO storage of 12 months. For the following tags, there is an additional “short” scanning interval of 5-seconds, with FIFO storage of two weeks:
   Current A, Current B, Current C, Voltage A-B, Voltage B-C, Voltage C-A, Power Factor Total, Apparent Power Total, Reactive Power Total, Real Power Total, and Frequency.

   For instructions on changing the “short” scan interval settings, see *Trend Tag Scan Intervals* on page 134.
10. The Driver Parameters box allows you to specify certain parameters to be attached to device profiles. Currently used in IEC61850 devices, the available parameters will automatically populate this box. See the illustration below for an example.




    In this example, Device 2 has two parameters, DataSets and Report Control Blocks.
11. Check the parameter(s) that you want to include in this profile.

    To edit, this parameter, return to the Create Device Profiles tab, and click the Parameters sub-tab. See ..... for information on editing parameters.
12. Check the Close Wizard box, and click Finish to return to Create Device Profiles tab/ Or, leave it unchecked, and click Finish to return to the Add/Edit Device Profile screen.

© 2011 Schneider Electric All Rights Reserved

# G3200 Device Setup

Use these instructions to set up a G3200 gateway in PowerLogic SCADA.

## For use with multiple devices

Before you begin, create the ICD files for each unique device type that will communicate via the G3200.

### In the Profile Editor

1. Import each unique ICD file.
2. Create the profiles for each device.
3. Modify existing profiles, as needed (adding/modifying tags, etc.)
4. Create the project that will include the GS3200 (mark profiles under the G3200 as data concentrated devices).
   a. Ensure that "Add As Default" is not checked for the project.
   b. Add the first device profile.
   c. At the Select Profiles screen, enter the Configured Name, and check the "Data Is Concentrated" box.
   d. Continue with steps "b" and "c" for additional device profiles.
5. Run the PowerLogic SCADA and to SCL exports.

### In SFT850

Create the CID file for the G3200 gateway.

### In PowerLogic SCADA

1. Open the Profile Wizard.
2. To the System Devices, add an IEC 61850 data concentrator for the G3200.
   a. From the Enter instance information screen, select the BRCBs that you need.
   b. Select the CID file you created in SFT850.
   c. Complete the remaining steps in the Profile Wizard.
3. Add a new device for each device under the G3200.
   a. From the Enter instance information screen, change the logical device as needed. Select the unit name of the G3200 device for the data concentrator.

## For use with a single device

Although we recommend that you add individual G3200 devices as described in the section above, you can also do it this way:

Before you begin, create the ICD files for the device type that will communicate via the G3200.


© 2011 Schneider Electric All Rights Reserved

### In the Profile Editor

1. Import the ICD file.
2. Create the device profile.
3. Modify the profile as needed
4. Create the project, but do not mark it as Data Concentrated Devices.

### In SFT850

Create the CID file for the G3200 gateway.

### In PowerLogic SCADA

1. Open the Profile Wizard.
2. Add the device.
   a. From the Enter instance information screen, select the BRCBs that you need.
   b. Change the logical device as needed.
   c. Select the CID file you created in SFT850.
   d. Complete the remaining steps in the Profile Wizard.

## IEC 60870-5-104 Protocol Support

NOTE: PLS_Include and genies may not be completely compatible with IEC 60870-5-104 devices. Be sure to properly test your system when using the IEC 60870-5-104 driver.

PowerLogic SCADA does not include IEC 60870-5-104 support in its install. However, you can add this protocol to a project. To include the IEC 60870-5-104 driver in your project, you must obtain and add a license, install the driver, and run a support utility to enable Profile Editor support. Follow these steps:

1. To obtain the license key, contact your local sales representative. Add the license to the dongle. If you need more information, see CiUSAFE dialog properties in the citectscada.chm help file.
2. The IEC 60870-5-104 driver is included in the install executable: Extras\Drivers\Specialty Drivers\IEC870-5-104.
3. Run the Profile Editor support utility. The .exe file is in the Program file directory: Profile Editor\v7.20\Add IEC 104 Driver Support.exe.

You can now use the IEC 60870-5-104 protocol. In the Profile Editor, you can enter addresses (though the Profile Editor will not verify that the format is correct).

## DNP3 Protocol Support

You can create device types and profiles that use the DNP3 protocol. See *Adding a DNP3_TCP Device* on page 101 or *Adding a Serial Device to a PowerLogic SCADA Project* on page 99 for more information.

You will then be able to enter DNP3 addresses, although the Profile Editor will not verify that the address has an allowed format.

© 2011 Schneider Electric All Rights Reserved

The Profile Editor will include device types, and will include profiles for ION 7650, which natively supports DNP3. The Profile Editor will include device types for Sepam 20, 40, and 80 that have the ACE969TP module (which supports DNP3).


© 2011 Schneider Electric All Rights Reserved

# Setting Up Projects in the Profile Editor

The Set Up Projects tab and screen are used to create separate projects for each customer or installation. This tab makes it easy to select only the devices that are used at that site. Project data is exported to PowerLogic SCADA for use in the Device Creation Wizard.

To access these screens, click the Set Up Projects tab on the main Profile Editor screen.

In this section, you can learn about:



## Project Screens and Workflow

This section contains two screens:

The **Set Up Project tab** has three sub-tabs:

- The Selected Device Profiles tab displays all of profiles that are included in the Project that is displayed in the drop-down menu.
- The Customize Tag Names sub-tab provides a means of customizing individual tag names.
- The Project Parameters sub-tab allows you to add optional lines of information about this project. This information will be exported and can be used for verification or identification when you want to import the project (for example, you might add a version number or creator name).

You can click Export Project to create an .XML file that contains all of the project data necessary for use in the Profile Wizard. If PowerLogic SCADA is installed and the corresponding PowerLogic SCADA project has been created, this also copies the file that is used by the Device Creation Wizard to the PowerLogic SCADA project.

On the **Add/Edit Project screen**, you can add, edit, or delete projects.

### Typical Workflow

To create a project file, you must first have established tags, device types, and device profiles. Additionally, you need to set up at least one base unit/conversion template. After these files are created, follow these steps:

1. Open the Add/Edit Project screen.
2. Add a new project, or copy and edit an existing project.
3. Select the device profiles that you want to use for this installation.
4. If a device profile has multiple drivers, choose the driver, and determine whether the individual device types will use functional addresses and act as data concentrators.
5. Save the project and close the Add/Edit Project screen.
6. To customize tag names: From the Set Up Project tab, click the Customize Tag Names sub-tab. Change the name of any tag. For example, the customer might need “Current A” to read “Current Phase A.” The customized tag name will be used

© 2011 Schneider Electric All Rights Reserved

in all device profiles in the project for which you have created the customized tag. For this change to be in the PowerLogic SCADA project: you need to delete the device profile from the PowerLogic SCADA project and then re-export it.

7. To add optional project information: From the Set Up Project tab, click the Project Parameters sub-tab. You can add optional information that will help verify or identify this project later. You could, for example, the version or the creator's name. This information will be available when you import this project at a later date.
8. To refresh tags: From the Set Up Project tab, click the Selected Device Profiles sub-tab. Click the Refresh Tags button for any profile. You are prompted to confirm that you want to update changes you have made to this tag for the selected profile. For this change to be in the PowerLogic SCADA project: you need to remove the device profile from the PowerLogic SCADA project and then re-add it.
9. Click Export Project to create an Equipment.Profiles file of all of the profiles included in the project. To view Equipment.Profiles, click the folder button, to the right of the Export button:

## The Set Up Projects Tab

Use the Set Up Project tab to begin adding, editing, or deleting projects. A project includes all of the tags that belong to the device profiles that you have created and added to the project. From this screen you also export individual projects to the .XML file format that can be added to PowerLogic SCADA (using the Profile Wizard).

This screen includes three tabs:

- Selected Device Profiles: (read only) You can view all of the profiles that are included in each project in the system. Profiles are listed with their descriptions.
- Customize Tag Names: You can customize tag names (for example, instead of Current A, you might need to use Current Phase A) within a single project. See *Customizing Tag Names* on page 68.
- Project Parameters: You can add optional information to be associated with the export. This information can help you identify the correct project when you are importing. See Adding Project Parameters.

To add or edit project information, click Add/Edit. The Add/Edit Project screen displays. See *Adding, Editing, or Deleting a Project* on page 66 for more information.

To export a project, select it from the Project drop-down menu, choose the export option you want (File > Export), then click Export. See *PowerLogic SCADA Export* on page 69 for more information.

To specify a tag to be used for onboard waveform captures, click the Onboard Alarm Tags sub-tab. Locate the tag and check the Waveform box.

To view the most recently exported project, click the folder button to the right of the Export button:

## Adding, Editing, or Deleting a Project

Use the Add/Edit Project screen to begin adding, editing, or deleting projects. A project includes all of the tags that belong to the device profiles that you have created and


© 2011 Schneider Electric All Rights Reserved

added to the project. From this screen you also export individual projects to the format that can be added to PowerLogic SCADA (using the Profile Wizard).

### Adding a Project

To add a project:

1. First ensure that you have set up the tags, device types, and device profiles that you want to include. Also, add at least one unit template.
2. Click the Set Up Projects tab, and click Add/Edit.
3. In the Project Options box, click Create New or Create From.

   If you are creating from another project, choose it from the Project to Create From drop-down list.
4. Type a Project Name: maximum 32 characters; name must be alpha-numeric only, must begin with an alpha character, no punctuation allowed. Do not use:

   \ / : * ? < > |
5. To view a list of projects that have already been added to PowerLogic SCADA, click the Display Projects button:

   A list displays with the projects that have been added to PowerLogic SCADA (greyed out if there are no projects yet or if the Profile Editor is not on the same computer as the server). To open a project for editing, select it and click OK.
6. If you want to lock the project, preventing anyone from editing it, check the Lock this Project box. This action cannot be undone. If you wish to edit a locked project, you must use the Create From feature to add a new one, then delete the locked one.
7. Type a Description for the project. This description displays as a tool tip when you hover over the project name on the main Set Up Project tab.
8. Select a Unit Template from the drop-down list. Unit templates are created on the Units screens. See *Setting Up Engineering Templates and Selecting Conversions* on page 28 for instructions on creating templates.

   To add a new unit template, click Set Up Eng. Unit Templates. The Set Up Engineering Unit Templates page displays. See *Adding or Editing a Base Engineering Unit or Conversion* on page 30 for help.
9. From the Device Profiles box, select the first profile you want to include in this project.

   If this device profile will NOT have functional addressing or data concentration, check the "Add As Default" box at the bottom of the screen. (For a description of functional addressing, see the Functional Addressing entry in *Appendix 4: Glossary* on page 196.)

   Click  to move the device profile to the Selected Device Profiles box.

   If the Select Profile Drivers screen displays, one of the following is true.

   — You did not check the "Add As Default" box for a device type, so the system does not know how to use the functional address/data concentrator option. Check the appropriate box to turn the related option "on."

   — At least one of the device types in this profile includes multiple drivers. For each multiple-driver device type listed, choose the driver that you want to use in this project. Additionally, you can check either of the Functional Address or Data Is Concentrated boxes to enable those features.

© 2011 Schneider Electric All Rights Reserved

10. Give the device type a Configured Name. This name might indicate its status (which driver it uses, whether it has a functional address, etc.) in future project references.
11. When all profiles are added, click Save to save the changes, or click Save & Exit to save changes and close the screen.

### Editing a Project

You can only edit projects that are unlocked.

To edit a project:

1. Open the Add/Edit Project tab (from the Set Up Project tab, click Add/Edit).
2. In the Project Options box: Select the project to be edited from the drop-down list, then click Edit Existing.
3. You can change any attribute of the project.
4. Click Save to save the change, or click Save & Exit to save changes and close the screen.

### Deleting a Project

You can only delete unlocked projects.

To delete a project:

1. Open the Add/Edit Project tab (from the Set Up Project tab, click Add/Edit).
2. In the Project Options box: Select the project to be deleted from the drop-down list, then click Delete Existing.
3. Click Delete.

## Editing and Deleting Information in a PowerLogic SCADA Project

After you have exported a project to a PowerLogic SCADA project, you still need to use the Device Creation Wizard to add system information to a PowerLogic SCADA project. See *Before You Use the Profile Wizard* on page 93 for information about this process.

## Customizing Tag Names

From the Set Up Project tab, click the Customize Tag Names sub-tab. You can add a custom name for any tag in the system, predefined and custom tags. The customized name will be used anywhere the original name would be used, but only for the project that is selected in the drop-down menu. When you use the Export option, it will be used by the Profile Wizard.


© 2011 Schneider Electric All Rights Reserved

# Importing and Exporting Project Files

In the Profile Editor, you can import and export these files:

- Export all of the tags and devices from a Profile Editor project into a PowerLogic SCADA project; see *PowerLogic SCADA Export* on page 69.
- Export SCL files, which allows you to export IID files that have been previously imported from an SCL file. The IID file can then be imported into other instances of the Profile Editor. See *SCL Export* on page 71.
- Export a Profile Editor project. This makes a backup copy, which you can later import into a different instance of the Profile Editor. This is useful when you want to share custom tags and devices. See *Profile Editor Export* on page 71.
- Import a project from another instance of the Profile Editor.
- Import SCL files. This allows you to import from the profile data of IEC61850-compliant devices and create device types. These files can be exported as an IID profile or as a PowerLogic SCADA profile.

  The import process works the same for each type of import. The only exception is that you cannot import profiles when you are importing SCL files. See *Importing Files into the Profile Editor* on page 72.

  When importing data, you will need to reconcile the import information with the information that exists in the Profile Editor.

  You can also use templates, both in exporting and importing. See *Using Import Templates* on page 76.

## PowerLogic SCADA Export

Use this feature to export a project from the Profile Editor into PowerLogic SCADA.

If the Profile Editor is on the same computer with PowerLogic SCADA, and if you have created a matching project in the PowerLogic SCADA project, this process will copy all project data—device tags, device types, and device profiles—into the PowerLogic SCADA project. If the Profile Editor is not on a computer with PowerLogic SCADA, you need to manually move the exported file to the PowerLogic SCADA server. See **Moving Files if the Profile Editor is not with the Server**, below.

### Before you export

If you are exporting a project for the first time to the PowerLogic SCADA project, you need to create a matching project in PowerLogic SCADA. To do this:

1. Launch the instance of PowerLogic SCADA to which you will be exporting data: Click Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Explorer.
2. From the Citect Explorer window, click File > New Project.
3. In the New Project Window, add the project . Be sure that Template Resolution is SXGA.

   If you have questions about any of the fields, click Help.
4. Add your project to the Profile Editor, ensuring that the name matches exactly the one that you added in PowerLogic SCADA (to ensure that it correctly exports to its matching project).

### To export a Profile Editor project to the PowerLogic SCADA project:

1. Open the Set Up Projects tab.
2. From the Project list, select the project to be exported.
3. Click File > Export; then check the PowerLogic SCADA Export option. (The selected export(s) are indicated beneath the Export Project button.)

© 2011 Schneider Electric All Rights Reserved

4. Click Export Project.

   NOTE: If you have added custom tags to devices, but the tag addressing is incomplete, a message displays with the name(s) of the device profile(s) that contain the tag(s). Return to the Define Device Type Tags tab. Locate any tags for which “Edit...” is red. Click Edit... to open the Edit Address screen. Make the necessary changes. From the Set Up Project tab, refresh the tags for those profiles. Then try exporting again.

   A progress bar displays while the various profiles are saved, The resulting files are exported to these locations in the Profile Editor (assuming that you accepted the default locations during installation):

   — Each Project file, used by the Profile Editor, is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.20\Projects (for Vista operating systems, it is ProgramData\Schneider Electric\Profile Editor\v7.20\Projects).
   — Each Profile Wizard profile file is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.20\WizardProfiles\[project name] (for Vista operating systems, it is ProgramData\Schneider Electric\Profile Editor\v7.20\WizardProfiles\[project name]). A single file for each included profile.
   — The Equipment.Profiles file (contains all of the Profile Wizard profile information and the base profile information used by the Profile Wizard) is stored in Program Files\Schneider Electric\Profile Editor\v7.20\WizardProfiles\[project name]\Profile Wizard.

   Both the Application Data directory (on Windows XP) and the ProgramData directory (on Vista) are hidden folders by default.

   In PowerLogic SCADA, files are located in the following folders:

   — DeviceProfiles contains .XML files for every profile (these are used by the Profile Editor).
   — DeviceTypes contains .XML files for all device types (these are used by the Profile Editor).
   — Projects contains all .XML files for all projects (these are used by the Profile Editor).
   — DeviceWizardProfiles contains the exported device profiles and equipment profiles files, organized by project (these are used by the Profile Wizard).

5. On the Project Editor window, use the Profile Editor to add device information.

### Moving Files if the Profile Editor is not with the Server (Reusing a Project)

When the Profile Editor is not on the same computer as the PowerLogic SCADA server, you need to move the export file to the server computer. Follow these steps:

1. Export the project from the Profile Editor.
2. Click the folder icon beside the Export link:

3. Copy the file (Equipment.profiles) that displays and move it to a portable drive.
4. On the PowerLogic SCADA server computer, paste Equipment.profiles in this location:

   [Drive Letter]:\Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[Project] (for Vista operating systems: [Drive Letter]:\ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[Project])

   where:

   [Drive Letter]: is the drive on which you installed the PowerLogic SCADA server


© 2011 Schneider Electric All Rights Reserved

the Application Data and ProgramData folders cannot be hidden (set the folder view for “view hidden folders”)

[Project] is the name of the project you are creating; you must have already added this project to PowerLogic SCADA (see Before You Export, above).

5. From the Citect Project Editor, click Tools > Profile Wizard to begin adding device information to the PowerLogic SCADA project.

## Profile Editor Export

To back up a PowerLogic SCADA project file, see *PowerLogic SCADA Backup* on page 129.

Use this procedure when you want export (back up) a Profile Editor project for re-use in another instance of the Profile Editor. This is useful when you have custom tags and custom devices that you want to share in other projects. After you export the project, you can use the Import feature to apply this project to another version 7.20 Profile Editor project.

1. From within the Profile Editor, click the Set Up Projects tab.
2. From the Project drop down box, select the project you want to back up.
3. Click File > Export, then ensure that the Profile Editor Export option is checked.
4. See *Customizing Tag Names* on page 68 and Adding Project Parameters for the information you need to make the changes that you want.
5. Click Export Project.

Exported projects will include, in addition to the project data:

- a unique project name, the date of the export
- the name of the computer to which it was saved
- the (optional) description added when the source project was created.

The project (which will be named YOUR PROJECT.pls) is exported to the following location:

C:\Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.20\WizardProfiles\YourProjectName\Profile Editor Export\YourProjectName.pls.

## SCL Export

Use this feature to export IID files (previously imported from an SCL file). The IID file can then be imported into other instances of the Profile Editor.

This process does not correct any errors in the files. If the imported file was an IID file from a different instance of the Profile Editor, it will contain the same configuration and communication information as the original. If the imported file was a Gateway SCL file with multiple devices, you can export each device as a separate IID file (the configuration and communication information is taken directly from the Gateway SCL file).

The only ways you can edit these files are:

- You can delete data sets, and then add new ones.
- You can edit report control blocks (buffered or unbuffered).

Perform these edits in the device profile before you export, and they will be exported to the IID file.

© 2011 Schneider Electric All Rights Reserved

### Exporting the File

Follow these steps to export IID files:

1. From the Set Up Projects tab, select the project from which you want to export. (The project must have devices that include ICD files.)
2. Click File > Export; then check the SCL Export box.

   NOTE: The export(s) that you select display beneath the Export Project button, on the right side of the screen.
3. Click Export Project.

The Export Summary displays with the results of the export. When the export displays under the Success topic,the listed files were exported. When the export displays under the Warnings topic, the reason for the failure to export is listed for the device types shown.

The exported files, listed according to their device types, will be saved in:

C:\Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.20\WizardProfiles\<project name>\SCL Export\sclFileName.iid).

## Importing Files into the Profile Editor

Use this feature to import either an existing project or SCL files into the Profile Editor. This is commonly used to share project information by importing custom tags and devices from another instance of the Profile Editor; but you can also import SCL files from an IEC 61850 device.

For specific information about importing SCL files, see *Importing SCL Files* on page 76

For Profile Editor projects, you can import tags, device types, and profile. For SCL imports, you cannot import profiles.

NOTE: You cannot complete the import until you match, merge, or reject every item.

Follow these steps to import data into the Profile Editor:

1. Before you begin, consider the source of the information you want to import. We strongly recommend that you use a master PC from which you draw this information. This will ensure that you are using a single source. Also, back up your data folder before you start. This gives you data to revert to, in case you accidentally lose data.
2. Note the location of the project file or SCL files that you want to import.
3. From the Profile Editor, click File > Import.
4. At the Import File Selection window, click the browse button (  ), then navigate to the location of the file you want to import.

   The Import Properties box displays the _ProjectName, _Description, _DateTime, and _ComputerName information. These lines were automatically generated for this file. Any additional lines will be information that you added on the Project Parameters sub-tab when you created or exported the project.

   Use this information to verify that you are about to import the files that you want.
5. If desired, select an import template from the drop down list. (To create a template, see Creating a New Template During Import in *Using Import Templates* on page 76 .) If you select a template, the import will accept default properties from the template. For example, if the template has alarm settings from a device, and you are importing tags for that device, the import will use those alarm settings.


© 2011 Schneider Electric All Rights Reserved

6. When you are sure you have the correct file, click Start Import.

   The system analyses the import and attempts to match imported items with existing items on the local machine. The Reconcile Import Screen displays, allowing you to begin the process of importing or rejecting individual tags.

   See *Import Reconciliation Screen (Search)* on page 73 for a description of the parts of this screen.

7. As you view the Import Reconciliation screen, you work through the items in the left-hand tree. You respond to each item according to your preference for it. You must match, merge, or reject first units, then tags, and finally the device type.

8. For each item in the left-hand pane, when you match it with a local item, the right-hand pane changes to allow you to edit how you want the match to take place.

   NOTE: You can click Unmatch at any point, to undo a match that you have made, but not saved.

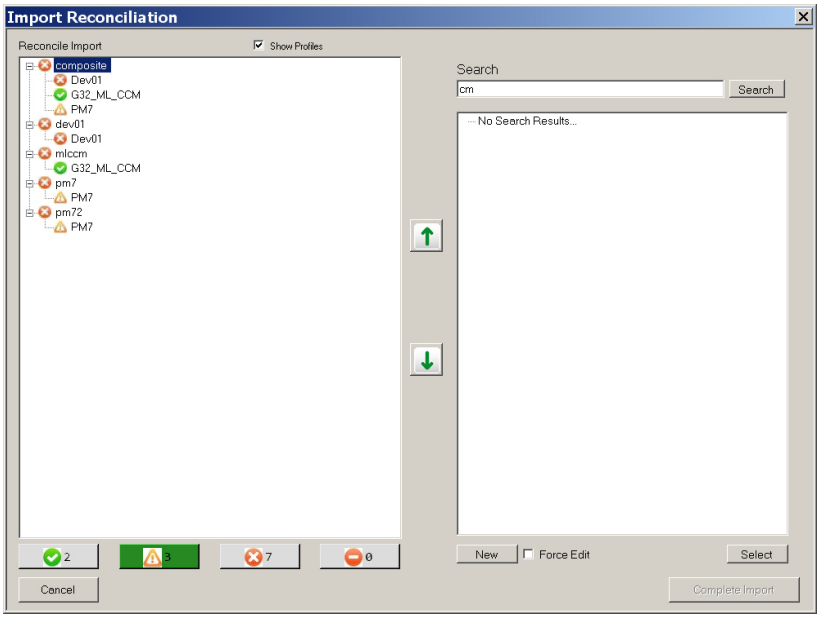
   See *Import Reconciliation Screen (Merging-Matching)* on page 74 for instructions on responding to each type of tag.

9. **After you have matched. merged. or rejected all items in the import list, the Complete Import button becomes live. Click Complete Import**.

   After the import is saved, the Save Import Template dialog displays. See *Using Import Templates* on page 76 for instructions on creating, using, and deleting import templates.

## Import Reconciliation Screen (Search)

Use the Reconcile Import Screen in Search mode to begin finding matches for the items you are importing. The following image illustrates an import with exact matches ( ), partial matches ( ), and no matches ( ):




The parts of this screen are:

**Left-hand pane**:

- The tree view presents the imported file's hierarchy: device profiles, device types, tags, and units. Each item has an icon to its left, indicating the system's initial assumption about how this item is matched. You can accept or override these assumptions.

© 2011 Schneider Electric All Rights Reserved

- The Show Profiles box controls whether the tree view lists device profiles (it is possible that device profiles will be discarded during the import).
- At the bottom of the page, the four buttons control how the tree navigates. (Each individual item also has one of these icons beside it, indicating its status.) You can select the items singly or in combination (click Ctrl +). The buttons are:

   : exact match; item is either a perfect match to a local item, or you accepted a merge for it.

   : partial match; item is a partial match to a local item.

   : no match; item does not match any local item.

   : ignore this item; you chose to ignore this item in this import.
- Cancel: Click to back out of the import without saving any of your selections.

**Right-hand pane**:

- Up and Down arrows: Use to navigate through the list of items in the left-hand pane, based on the button selected. For example, if you highlight the "no match" icon, and then click the down arrow, the selection in the left-hand pane will move down to the next "not match" item.
- Search box: When you select an item from the left-hand pane, type a search term (such as "power") to display a list of possible matches.
- Search results: The large box is populated with the possible matches, determined by the search word(s) you enter. They are listed in descending order of how well they match the attributes of the item.
- New: Click this button to add the selected import item into the project. It will keep all of its attributes, and it will be changed in the tree view to show it is an exact match.
- Force Edit: When all of the data is present for the item, but you still want to edit it, check this box to open the window required to edit it (e.g., Create Device Type window).
- Select: Click to match an import item with a selected item in the search results.

### Editing the Item

If you have checked Force Edit, a screen displays for you to edit the item's information. This allows you to make changes to an item, even though it may be an exact match with a local item. This new information will be applied to the item after you complete the import.

### Completing the Import of an Item

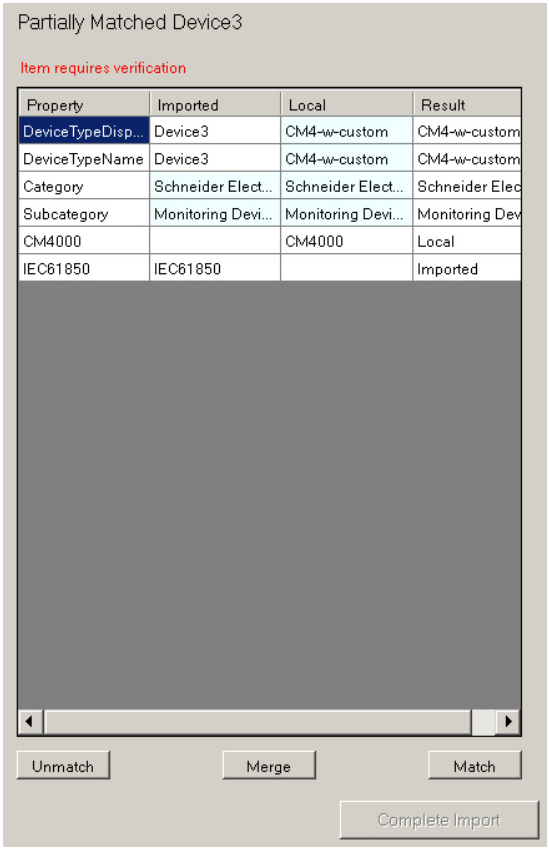
When you make a match—exact or partial—between an imported item and a local item, the right pane of the screen displays information about the match. See *Import Reconciliation Screen (Merging-Matching)* on page 74 for more information.

## Import Reconciliation Screen (Merging-Matching)

The right-hand pane of the Import Reconciliation screen changes after you match an import item with a local item.

The following example illustrates the screen when there is a partial match:


© 2011 Schneider Electric All Rights Reserved




The parts of this window are:

- Title: Indicates which of the three "states" the import item is in: matched (the item name displays), partially matched (you will need to merge the two items or accept the match that is in the final column), or merging (the system assumes a match in the final column, but you can edit each attribute). These actions to take for these three states are described below.
- The columns describe the imported vs local items:

  Property: The property for which the other columns provide definitions

  Imported: The value of the item in the import file

  Local: The closest local match for the imported item

  Result: The item as it will be added in this import
- Unmatch: Removes the match for the item; if you were merging, the merge grid goes back to its original state for you to either reject it, find a match, or import as new.
- Merge: When the import item is matched with an unlocked item, click here to begin merging their attributes. The "Merging" heading displays, and you can begin choosing the setting to be used for each attribute.

  The shaded items are the ones assumed by the system; by default, each one will be the Local setting. To change an item from local to import, double-click the item name in the Imported column. Notice how the Result column changes to reflect the name change. To toggle back to the Local setting, simply double-click the name in the Local column.
- Match: Accepts the attributes of the local item for the new imported item (item will turn to a green check). You can use this to accept an item without changing it or after merging its attributes.

© 2011 Schneider Electric All Rights Reserved

- Complete Import: **This button becomes live only after you have matched, merged, or rejected all items in the left-hand pane**.It first alerts you that this operation cannot be undone. If you accept that, the import is final; new data is saved to the local computer. This is permanent; you cannot undo it.

## Importing SCL Files

You can import SCL files from individual devices, provided the files conform to IEC61850 specifications. You can also import an individual device from a Gateway SCL file that contains multiple devices.

NOTE: You can only import SCL files that meet the schema requirements for Ed 1.4 of IEC61850. If an SCL file does not meet these requirements, an error message will display, telling you that the scheme must validate against the scheme of Ed. 1.4. The Profile Editor will accept SCL files that use either Ed. 1 or Ed. 2 data structures; but it will apply data structures only as defined in Ed. 2.

During this import, you will need to reconcile mismatches; and data will be available for creating SCADA device types, device profiles, and projects. If you import an SCL for a PM700, note that all tags for date and time are excluded by default.

You can save the information in one of two ways:

- **IID file**. This IID file will maintain all of the configuration and communication information that comes from its device. The only items you can change are:
  - — You can delete datasets and control blocks, and add new ones.
  - — You can edit buffered and unbuffered control blocks (provided you have created them in the Profile Editor).
- **PowerLogic SCADA profile**. The data will then follow the normal rules for the profiles in this project.

See *Importing Files into the Profile Editor* on page 72 for instructions on importing.

## Using Import Templates

You can create, edit, and apply templates when you import files. You can also delete import templates. A template will include tags that you import into the Profile Editor from a project in another instance of the Profile Editor, or it will contain ICD files from an IEC 61850 device.

You will want to create a template for custom situations, like when you are importing SCL files or adding custom tags and devices.

### Creating a New Template During Import

To create a new template:

1. From the either type of import (Profile Editor or SCL), choose the file (.pls or .icd) that you want to import.
2. Complete the matching for the items.
3. Click Complete Import.
4. At the Save Import Template prompt, click Yes.
5. Click the New radio button, then type a name for the new template. The name must begin with a letter. It can contain alpha-numeric characters, as well as dashes and spaces. Click OK to save it.


© 2011 Schneider Electric All Rights Reserved

In future imports, you will be able to apply this template. When you do, the system will automatically match, where appropriate, the import items with the local items.

### Applying a Template During Import

In this procedure, you will import files, and you will either create a new import template, or you will edit an existing one.

NOTE: Be careful when applying a template: you will overwrite an existing template on the local computer with the information that you choose during matching. Once completed, you cannot undo this.

To apply an existing template:

1. From the Set Up Projects tab, select a project for which you want to import data.
2. Click File > Import, then choose the file (.pls or .icd) that you want to import.
3. From the Import Template drop down list, choose the template you want to use. This is just a starting point for this import to make it quicker to match items. You will apply the template in step 7.
4. Click Start Import.

   After the import completes, the Import Reconciliation screen displays. The list in the left-hand pane should have some exact and partial matches.
5. As you work through the items, you must either designate that each a match or ignored.
6. When all items are completed, click Complete Import.
7. At the Save Import Template dialog, click No to import without applying a template. Or click Yes to either save a new template or edit the one you chose in step 3:
   - To create a new template for this import, click New, then type an Import Template Name.
   - To edit a template, click Edit, then select the template from the drop down menu. This will edit the template by adding the changes you made during matching. This cannot be undone after you click OK.
8. Click OK.

The import is completed, and the new template is created, or the existing template is edited to include the changes you made during matching.

### Deleting a Template

You can delete any import template, even if it was applied during a previous import.

To do this:

1. Click Settings > Remove Import Templates.
2. At the Import Templates dialog, select the template you want to delete, then click Delete.

The template is deleted.

© 2011 Schneider Electric All Rights Reserved

# Understanding Tags

PowerLogic SCADA includes a variety of tag types: real-time, alarm, and trend. Most of the tags that you will need are already added. However, you can add custom tags to suit special needs. This section describes how tags are constructed and provides further specific information about the construction of format codes, logic codes, and addresses.

In this section, you can learn about:



## IEC61850 Tag Construction

IEC61850 tags have a great deal of flexibility, which allows them to specify the way that functions are implemented in devices. The IEC61850 tag was developed for medium-voltage and high-voltage applications, such as monitoring, control, and substation automation.

Some of our devices include data and functionality that are not yet covered by IEC61850. For these devices, the general IEC61850 formatting was followed when creating tags.

If you are writing Cicode (see *Using Cicode to Customize a Project* on page 132). You will need to know the IEC61850 tag name that you added to the device profile for that device. You can print the .csv file to view tag names (see *Printing the .CSV File* on page 39). Apart from that, you would only need to add tags if you are installing a third-party device that is not standard to PowerLogic SCADA. If you do need to add tags, create any category you wish, and follow the format shown below.

Tags shall be no longer than 29 characters. Use a backslash as a separator between tag parts. Tags are constructed in this manner:

EquipmentName\Logical_Node\Data Object\Data Attribute (may have more than one)

The following table lists the main categories for the common logical nodes. After the table, the most commonly used category (Mxxx: metering and measurement) is described.

| Category Name | Description |
|---|---|
| Axxx | automatic control; e.g., ATCC (tap changer), AVCO (voltage control) |
| Cxxx | supervisory control; e.g., CILO (interlocking), CSWI (switch control) |
| Gxxx | generic functions; e.g., GGIO (generic I/O) |
| Ixxx | interfacing/archiving; e.g., IARC (archive), IHMI (HMI) |
| Lxxx | system logical nodes; e.g., LLNO (common), LPHD (physical device) |
| Mxxx | metering and measurement; e.g., MMXU (measurement), MMTR (metering), MSTA (metering statistics), MSQI (sequence and imbalance), MHAI (harmonics and interharmonics) |
| Pxxx | protection; e.g., PDIF (differential), PIOC (instantaneous overcurrent or rate of rise.), PDIS (distance), PTOV (time-overvoltage) |
| Rxxx | protection related; e.g., RREC (auto reclosing), RDRE (disturbance) |
| Sxxx | sensors, monitoring; e.g., SARC (arcs), SPDC (partial discharge) |


© 2011 Schneider Electric All Rights Reserved

| Category Name | Description |
|---|---|
| Txxx | instrument transformer; e.g., TCTR (current), TVTR (voltage) |
| Xxxx | switchgear; e.g., XCBR (circuit breaker), XCSW (switch) |
| Zxxx | other equipment; e.g., ZCAP (cap control), ZMOT (motor) |

The following example illustrates the IEC61850 tag for current A:

EquipmentName\MMXU1\A\PhsA

where:

M = the category

MXU = measurement of currents, voltages, power, and impedances

1 = the instance (there could be multiple MMXU tags)

A = the data object, current

PhsA = the attribute that further defines the data object, phase A

All of the tags that are currently used in the system can be viewed from the Profile Editor, Define Device Type Tags tab. Click Settings > Display Advanced Properties to display the full tag names.

## Defining an Enumeration

An enumeration is a single value (0-15) that is used to define a condition that is determined by multiple-bit input. You will add enumerations to handle scenarios that are more complicated than simply true-false, to allow for dynamic contingencies. For example, when you need to use multiple bits to describe the position of a circuit breaker, you might do the following:

Bit *y* (closed) | Bit x (open). Note that the least significant bit is register 1.

| Bit x \| Bit y | Status | Circuit Breaker Position | Returned Value |
|---|---|---|---|
| 0 \| 0 | Indeterminate | Circuit breaker is neither open nor closed | 0 |
| 0 \| 1 | Open | Circuit breaker is open. | 1 |
| 1 \| 0 | Closed | Circuit breaker is closed. | 2 |
| 1 \| 1 | Error | Circuit breaker is reporting both open and closed condition. Possible device/wiring error | 3 |

Using the enumerated status, we place the register and bitmask for the open position in register 1 (least significant) and the register and bitmask for the closed position in register 2 (most significant).

## Using Special Tags to Control Circuit Breaker Status

When you want to include a device that does not have a pre-defined device profile (such as a third-party circuit breaker), you must identify the registers that the device uses for the operations you want, then choose the correct tags and tag addresses to write to these registers. Finally, when creating the one-line on the graphics page, you will choose the appropriate genie:

© 2011 Schneider Electric All Rights Reserved

1. Determine the device registers used for the open and close operations on the circuit breaker.
2. In the Profile Editor, choose the tag needed for each operation.
3. Ensure that tag address references the correct action and register(s). See *Editing Tag Addresses* on page 46 for instructions on editing the address,
4. When adding a genie for the circuit breaker on the graphics page, choose from the default library (see *Appendix 1: Default Genie Library* on page 163), or create a custom genie (see *Creating and Editing Genies* on page 123).

## Format Code Definitions

The address field is part of the tag. It includes a variety of attributes, some of which are required, and some optional. The following tables list the attributes, whether they are required, and their possible modifiers. All parts of a tag are case sensitive. The order of the fields is fixed; and all fields are separated by semi-colons. See *Logic Codes* on page 83 for templates of constructed tags.


© 2011 Schneider Electric All Rights Reserved

## Real-Time Format Code Definitions

| Attributes | Modifiers | Comments |
|---|---|---|
| T (type)<br>Required | SS = single status<br>DS = double status enumeration<br>ST = string<br>UT = UTC time<br>MV = measured value (float)<br>CM = complex measured value (float)<br>BC = binary counter (integer) | Temporarily, this may return a string; when PowerLogic SCADA is upgraded to handle large integers, this will change. |
| D (module—Micrologic devices) | B = BCM<br>P = PM<br>M = MM<br>C = CCM | |
| M/m/S/s/C/c/I/i (register type) | M = holding registers in hexadecimal<br>m = holding registers in decimal<br>S = input coil (status register) in hexadecimal<br>s = input coil (status register) in decimal<br>C = output coil (writable only) in hexadecimal<br>c = output coil (writable only) in decimal<br>I = input register (read only) in hexadecimal<br>i = input register (read only) in decimal | |
| Register Number Modifiers (register number from 1–4) | u## = ## registers are unsigned, ## is a decimal<br>s## = ## registers are signed; ## is a decimal | After the modifier, there may be a number indicating scaling factor. See "V," below in this table. Used for conversion to SI units, this number will be:<br>RegisterValue x scale<br>For SS and DS: there must be a 1U default; the modifier will be a bitmask:<br>- The mask must use hex only, 16 bits/register<br>- Attach the ones, then the zero mask, to the register; if you only have ones masks, just attach them<br>- Only one register cases can be inverted. Add **:I** after the masks for inversion. |
| N (scale) | numerical entries; range is -10 to 10 | N defines a constant scale; the logic code knows how to use it. |
| R (scale register) | the register number in decimal | R defines the holding register where the scale is held; the logic code knows how to use it. |
| E (priority) | single digit: 1, 2, or 3; default 2 is used if this is not included<br>(1 = high, 2 = normal, 3 = low) | Defines the priority PowerLogic SCADA uses in processing data. |
| V (conversion factor) | Use scientific notation without the decimal. | Examples: 354E-3 = 0.354<br>354E1 = 3540<br>Will be multiplied before the value is returned. |

© 2011 Schneider Electric All Rights Reserved

| Attributes | Modifiers | Comments |
|---|---|---|
| | | |
| L:P (logic code)<br>Required | The number that is used comes from the Logic Codes table. | L:P is the logic code for PowerLogic. Other codes may follow, such a L:I for ION.<br>For logic code descriptions, see *Logic Codes* on page 83 |

### Alarm Format Code Definitions

| Attributes | Modifiers | Comments |
|---|---|---|
| T (type)<br>Required | ALM = alarm | |
| D (module—optional for Micrologic devices) | B = BCM<br>P = PM<br>M = MM<br>C = CCM | BCM is straight addressing, and therefore, optional. |
| F (file)<br>Required | File number will be in decimal, up to 5 digits | |
| Q (unique ID)<br>Required | Unique ID will be in decimal. | This number can be huge. |

### Control Format Code Definitions Rules of Operation

These rules are true for predefined and custom codes:

| Address structure | Result |
|---|---|
| C:N;(action) | If 1, perform action. If 0, undefined. |
| C:N;(action1);(action2)<br>C:NO;(action1);(action2) | If 1, perform action1. If 0, perform action 2. |
| C:NC;(action1);(action2) | If 1, perform action2. If 0, perform action1 |

### Predefined Control Format Codes

| Attributes | Modifiers | Comments |
|---|---|---|
| C (command)<br>Required | NO = normally open<br>NC = normally closed<br>N = normal operation | Normal operation does not have a closed/open status. |
| OPERATE (command word) | n/a | Two required for NO and NC. |

### Predefined Reset Format Codes

| Attributes | Modifiers | Comments |
|---|---|---|
| Reset (command word) | n/a | Entering a one to this tag causes the reset to take place. |


© 2011 Schneider Electric All Rights Reserved

### Custom Control and Reset Format Codes

| Attributes | Modifiers | Comments |
|---|---|---|
| C (command)<br>Required | NO = normally open<br>NC = normal closed<br>N = normal operation | Normal operation does not have a closed/open status. |
| Followed by one or two entire “write” addresses; used only for logic codes 101, 102, 103. For logic code descriptions, see *Logic Codes* on page 83.<br>Write Address format: T:SS;m:##:#;L.:P:101<br>Example: C:NO;T:SS;m:1234:1;L:P101;T:SS;m:3456:1;L:P101 | | |

## Logic Codes

Logic codes tell the SCADA program how to mathematically operate on the values in device registers to give users the desired values. The following table lists each logic code with its related information.

© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| READS | | | | | | |
| Invalid Block (L:P:0) | IB | LONG | Up to 1,000 sequential registers | No | Generic only | Defines invalid blocks of memory in the device. The driver does not include these registers in block reads. |
| Date / Time (L:P:1) (3 register) | UT | LONG | 3 sequential registers | No | Generic – if it fits | **Register N**:<br>High byte = Month 1–12<br>Low byte = Day 1–31<br>**Register N+1**:<br>High byte = Year 0–199 (+1900)<br>Low byte = Hour 0–23<br>**Register N+2**:<br>High byte = minutes 0–59<br>Low byte = seconds 0–59 |
| Date / Time (L:P:2) (6 register) | UT | LONG | 6 sequential registers | No | Generic<br>- if it fits | **Register N**: Seconds 0–59<br>**Register N+1**: Minutes 0–59<br>**Register N+2**: Hours 0–23<br>**Register N+3**: Day 1–31<br>**Register N+4**: Month 1–12<br>**Register N+5**: Year 0–199 (+1900) |
| Date / Time (L:P:3) (3 or 4 register -- Circuit Monitor/ Power Meter) | UT | LONG | 3 or 4 sequential registers | No | CM/PM | **Register N**:<br>High byte = Month 1–12,<br>Low byte = Day 1–31<br>**Register N+1**:<br>High byte = Year 0–199 (+1900)<br>Low byte = Hour 0–23<br>**Register N+2**:<br>High byte = minutes 0–59<br>Low byte = seconds 0–59<br>**Register N+3**:<br>msec = 0–999 (unused) |
| Date / Time (L:P:4) (3 or 4 registers SEPAM) | UT | LONG | 3 or 4 sequential registers | No | SEPAM | **Register N**:<br>Bits 0–6 = Year: 0 –70 (2000– 2070)<br>71 – 99 (1971–1999)<br>**Register N+1**:<br>Bits 8-11 = Month<br>Bits 0-4 = Day<br>**Register N+2**:<br>Bits 8-12 = Hour<br>Bits 0-5 = Minutes<br>**Register N+3**: msec = 0-59,999<br>(seconds are ms/1000) |
| Date/Time (L:P:5) 3-register Micrologic | UT | LONG | 3 sequential registers | No | micro | **Register N**:<br>High byte = Month 1–12,<br>Low byte = Day 1–31<br>**Register N+1**:<br>High byte = Year 0–69 (+2000), Year 70–99 (+1900)<br>Low byte = Hour 0–23<br>**Register N+2**: |


© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| | | | | | | High byte = minutes 0–59<br>Low byte = seconds 0–59 |
| Date/Time<br>(L:P:6)<br>4-register Micrologic | UT | LONG | 4 sequential registers | No | micro | **Register N**:<br>High byte = Month 1–12,<br>Low byte = Day 1–31<br>**Register N+1**:<br>High byte = Year 0–69 (+2000), Year 70–99 (+1900)<br>Low byte = Hour 0–23<br>**Register N+2**:<br>High byte = minutes 0–59<br>Low byte = seconds 0–59<br>**Register N+3**:<br>msec = 0–999 (unused) |
| Date/Time<br>(L:P:7)<br>2-register Argos | UT | LONG | 2 sequential registers | No | Argos | The number of seconds since 01/01/2000 (00:00:00)<br>register 1 = MSB<br>register 2 = LSB |
| Modulo 10k<br>(L:P:10) | BC | STRING | Up to 4 registers | No | generic | Result is a string representation.<br>Range is 0 to 9,999,999,999,999,999<br>Each register has a range of 0 to 9,999<br>Result is:<br>– R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1 |
| Modulo 10k Val<br>(L:P:11) | BC | REAL | Up to 4 registers | No | generic | Result is a string representation.<br>Range is 0 to 9,999,999,999,999,999<br>Each register has a range of 0 to 9,999<br>Result is:<br>– R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Modulo 10k Energy<br>(L:P:12) | BC | STRING | Up to 4 registers | No | generic | Result is a string representation.<br>Range is 0 to 9,999,999,999,999.9<br>Each register has a range of 0 to 9,999<br>Result is<br>– (R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1)/1000 |
| Modulo 10k Energy Val<br>(L:P:13) | BC | REAL | Up to 4 registers | No | generic | Result is a string representation.<br>Range is 0 to 9,999,999,999,999.9<br>Each register has a range of 0 to 9,999<br>Result is<br>– (R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1)/1000<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |

© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| PL Digital Input SS (L:P:20) | SS | LONG | 2 registers | No | CM/PM | First register (100–199 inclusive) indicates that this is a digital input register.<br>Second register is masked to test for either one 1 or one 0.<br>Result is: 0 = off and 1 = on.<br>This result can be inverted. |
| PL Digital Input DS (L:P:21) | DS | LONG | 2 registers | No | CM/PM | Same as PL Digital Input SS except:<br>Result is 0 = intermediate, 1 = off, 2 = on, 3 = bad-state.<br>Inversion will invert only off and on states. |
| PL Digital Input TF (L:P:22) | SS | DIGITAL | 2 registers | No | CM/PM | Same as PL Digital Input SS except:<br>Result is: 0 = false and 1 = true.<br>This result can be inverted. |
| PL Digital Output SS (L:P:23) | SS | LONG | 2 registers | No | CM/PM | First register (200–299 inclusive) indicates that this is a digital output register.<br>Second register is masked to test for either one 1 or one 0.<br>Result is: 0 = off and 1 = on.<br>This result can be inverted. |
| PL Digital Output DS (L:P:24) | DS | LONG | 2 registers | No | CM/PM | Same as PL Digital Output SS, except:<br>Result is:<br>0 = intermediate, 1 = off, 2 = on, 3 = bad-state.<br>Inversion will invert only off and on states. |
| PL Digital Output TF (L:P:25) | SS | DIGITAL | 2 registers | No | CM/PM | Same as PL Digital Output SS except:<br>Result is: 0 = false and 1 = true.<br>This result can be inverted. |
| Status SS (L:P:26) | SS | LONG | Up to 4 registers | No | Generic | Each register is compared to a ones' mask. Optionally it can be compared to a zeros' mask. (Use the Edit Address screen in the Profile Editor to create masks for the user.)<br>Result is: 0 = off and 1 = on.<br>If there is only one register, the result can be inverted. |
| Status OR SS (L:P:226) | SS | LONG | 2 to 4 registers | No | Generic | Each register is compared to a ones' mask. These results are OR'ed together. Optionally, it can be compared to a zeros' mask. (Use the Edit Address screen in the Profile Editor to create masks for the user.)<br>Result is: 0 = off and 1 = on. |
| Status DS (L:P:27) | DS | LONG | Up to 4 registers | No | Generic | Same as Status SS except:<br>Result is: 0 = intermediate, 1 = off, 2 = on, 3 = bad-state.<br>Inversion will invert only off and on states. |
| Status OR DS (L:P:227) | DS | LONG | 2 to 4 registers | No | Generic | Same as Status OR SS except:<br>Result is: 0 = intermediate, 1 = off, 2 = on, 3 = bad-state. |
| Status TF (L:P:28) | SS | DIGITAL | Up to 4 registers | No | Generic | Same as Status SS except:<br>Result is: 0 = false and 1 = true.<br>This result can be inverted. |
| Status OR TF (L:P:228) | SS | DIGITAL | 2 to 4 registers | No | Generic | Same as Status OR SS except:<br>Result is: 0 = false and 1 = true. |
| Status Int (L:P:29) | BC | LONG | 1 register | No | CM/PM | One register is bit anded with one mask. The result will be an integer that can be used to choose the appropriate enumeration. |


© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| Status Enumeration (L:P:229) | EN | LONG | 1 to 4 registers | No | Generic | Each register is compared to a ones' mask. Optionally it can be compared to a zeros' mask. (Use the Edit Address screen in the Profile Editor to create masks for the user.)<br>Result is a combination of the results for each register, using this formula:<br>result for register 1 * 2^0 + result for register 2 * 2^1 + result for register 3 * 2^2 + result for register 4 * 2^3 |
| PL Analog Input (L:P:30) | MV/CM | REAL | 3 registers | No | CM/PM | First register (300–399 inclusive) indicates that this is a analog input register.<br>Second register is treated as a signed value.<br>Third register can contain a value from –3 to 3 and will be used to scale the second register (R2*10^R3).<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| PL Analog Output (L:P:31) | MV/CM | REAL | 2 registers | No | CM/PM | First register (400–499 inclusive) indicates that this is a analog output register.<br>Second register is treated as a signed value.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Scaled Register Signed (L:P:32) | MV/CM | REAL | 1 or 2 registers | Either (optional) | Generic | For a single register: treated as a signed value from –32,767 to +32,767. (-32768 will result in a NA)<br>For two registers: the registers will be concatenated together, the first register filling bits 16–32 and the second register filling bits 0–15. Values will range from<br>–2,147,483,648 to –2,147,483,647.<br>Values can be scaled using a fixed scale or a scale register.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Scaled Register Unsigned (L:P:33) | MV/CM | REAL | 1 to 4 registers | Either (optional) | Generic | For a single register: treated as an unsigned value from 0 to 65,535.<br>For two registers: the registers will be concatenated together, the first register filling bits 16–32 and the second register filling bits 0–15. Values will range from<br>0 to 4,294,967,295.<br>Values can be scaled using a fixed scale or a scale register.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Scaled Register Signed No NA (L:P:34) | MV/CM | REAL | 1 or 2 registers | Either (optional) | Generic | Same as Scaled Register except that a single register with value -32768 is acceptable and will be reported as such.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Scaled Register Signed SEPAM A (L:P:35) | MV/CM | REAL | 2 registers | Either (optional) | Generic | Same as Scaled Register except that 0xFFFFFFFF or 0x00007FFF will be NA.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |

© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| Scaled Register Signed SEPAM B (L:P:36) | MV/CM | REAL | 2 registers | Either (optional) | Generic | Same as Scaled Register except that 0xFFFFFFFF will be NA.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| IEEE 32 Real (L:P:37) | MV/CM | REAL | 2 sequential registers | No | Generic | Uses the IEEE standard for floating-point arithmetic (IEEE 754);<br>register 1 is MSB, register 2 is LSB<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Scaled Register Signed SEPAM 2000 Format B (L:P:38) | MV/CM | REAL | 1 register | Either (optional) | Generic | For a single register: treated as a signed value from –32,767 to +32,767:<br>From the value of the unsigned register, subtract 32768; then apply the scale.<br>0000 or FFFF will be NA.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| PL String (L:P:39) | ST | STRING | 1 to 10 sequential registers | No | Generic | Each register can represent up to two ASCII characters. |
| Sum Registers (L:P:40) | MV/CM | REAL | 1 to 4 registers | Either (required) | Generic | Result is:<br>R1 + … + Rn * 10^scale<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Divide Registers (L:P:41) | MV/CM | REAL | 3 registers | Either (required) | Generic | Result is:<br>R1/R2 * R3 * 10^scale<br>If R2 is zero, result will be #COM<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Multiply Registers (L:P:42) | MV/CM | REAL | 1 to 4 registers | Either (required) | Generic | Result is:<br>R1 * … * Rn * 10^scale<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Average Registers (L:P:43) | MV/CM | REAL | 1 to 4 registers | Either (required) | Generic | Result is:<br>Avg(R1 … Rn) * 10^scale<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Average Registers WF (L:P:44) | MV/CM | REAL | 2 to 4 registers | Either (required) | Generic | Result is:<br>Avg(R1 … Rn-1) * Rn * 10^scale<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Sum with Scale (L:P:45) | MV/CM | REAL | 2 registers | Either (required) | CM/PM | Result is:<br>(R1 * 10^scale) + R2<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |


© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| | | | | | | |
| Sum with Scale Unsigned (L:P:46) | MV/CM | REAL | 2 registers | Either (required) | CM/PM | Result is same as above, except unsigned.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Vector Math (L:P:47) | MV/CM | REAL | 2 registers | Either (required) | Generic | Result is:<br>sqrt (R1^2 + R2^2) x scale<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Vector Math IEEE (L:P:48) | MV/CM | REAL | 4 registers | Either (required) | Generic | Result is:<br>sqrt ([R1 R2]^2 + [R3 R4]^2) x scale<br>where [ ] indicates IEEE32 representation<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| CM4 Power Factor IEEE (L:P:50) | MV/CM | REAL | 1 register | No | CM4 | Returns the IEEE power factor. |
| PM8 Power Factor IEEE (L:P:51) | MV/CM | REAL | 1 register | No | PM8 | Returns the IEEE power factor (converted from IEC mode as necessary).<br>The device may be in IEEE or IEC mode if the device firmware version is 11.6 or higher. If the device firmware version is below 11.6, IEC mode is not supported.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| SP40 Power Factor IEEE (L:P:52) | MV/CM | REAL | 1 register | No | SEPAM 40 | Returns the IEEE power factor (converted from IEC mode).<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| ML Power Factor IEEE (L:P:53) | MV/CM | REAL | 2 registers | No | ML | Returns the IEEE power factor (converted from IEC mode as necessary).<br>The second input register must be the associated Reactive Power for the Power Factor requested.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Generic Power Factor (L:P:54) | MV/CM | REAL | 2 registers | No | Generic | R2/sqrt (R2^2 + R1^2)<br>where:<br>R2 = real power<br>R1 = reactive power<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Generic Power Factor - IEEE32 variation (L:P:55) | MV/CM | READ | 4 registers | No | Generic | [R3 R4]/<br>sqrt ([R3 R4]^2 + [R1 R2]^2)<br>where:<br>R3 = real power IEEE32 MSR<br>R4 = real power IEEE32 LSR<br>R1 = reactive power IEEE32 MSR<br>R2 = reactive power IEEE32 LSR |

© 2011 Schneider Electric All Rights Reserved

| Logic Code | IEC Type | PowerLogic SCADA Data Type | Register Definition | Scaled Required? (register or fixed) | Device Specific? | Notes |
|---|---|---|---|---|---|---|
| SP2000 Power Factor IEEE (L:P:56) | MV/CM | REAL | 1 register | No | SEPAM 2000 | Returns the IEEE power factor (converted from IEC mode).<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| WRITES (these are write-only; see below for Read/Write codes)<br><br>NOTE: If the device is capable of preventing (blocking) writes to its registers, verify that the "block" feature is disabled before you implement the write. | | | | | | |
| Status Write Register (L:P:101) | SS | LONG | 1 register | No | Generic | If you input 1 to this tag it will write the **MASK** value to the register. |
| Status Write Register AND (L:P:102) | SS | LONG | 1 register | No | Generic | If you input 1 to this tag it will read the register and **AND** the **MASK** with the register (This puts a 0 whereever there is a 1 in the mask and leaves the rest alone). |
| Status Write Register OR (L:P:103) | SS | LONG | 1 register | No | Generic | If you input 1 to this tag it will read the register and **OR** the **MASK** with the register (This puts a 1 whereever there is a 1 in the mask and leaves the rest alone). |
| Write Register Unsigned (L:P:110) | MV/CM | REAL | 1 register | Either | Generic | This will take the input value read in and divide out the scale factor and the conversion factor. It will then round to the nearest whole number and if it is a value from 0 to 65535 it will put this value in the register.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| Write Register Signed (L:P:111) | MV/CM | REAL | 1 register | Either | Generic | This will take the input value read in and divide out the scale factor and the conversion factor. It will then round to the nearest whole number and convert the signed value to an unsigned value from 0 to 65535. It will put this value in the register.<br>NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate. |
| READ/WRITES | | | | | | |
| Read/Write Holding Register (L:P:120) | MV/CM | LONG | 1 register | No | Generic | You can write any value from 0 to 65535 and read an unsigned value from the same register. |
| Read/Write Coil Register (L:P:121) | SS | DIGITAL | 1 register | No | Generic | You can write 0 or 1 and read a value from the same register. |

## Using Block Writes

Block writes represent blocks of registers that are updated in a single write operation. There are two types of block writes:

- **fixed**: fully specified and compiled before run time. Writing the value of '1' to such a variable tag causes the specified fixed values to be written to the specified registers.


© 2011 Schneider Electric All Rights Reserved

- **variable**: specified on the fly. The registers and the values to be written are not fixed; they are specified during run time by the user.

**Fixed block writes** have the following format:

```
T:BWF;[D:{B|C|M|P};]S:<start_register>,<values>
```

where

`B`, `C`, `M`, or `P` are applicable only to Micrologic devices (otherwise the D: section is omitted) and is the module (manager) identifier (Circuit Breaker, Chassis, Metering, Protection).

`<start_register>` is the first register number for a contiguous block of registers.

`<values>` is a comma-separated list of up to 10 values that will be written to the registers starting from <start_register>.

For example:

```
T:BWF;S:100,1,2,3,4,-5
```

**Variable block writes** have the following format:

where

`B`, or `C`, or `M`, or `P` is applicable only to Micrologic devices (otherwise the D: section is omitted altogether) and is the module (manager) identifier (Circuit Breaker, Chassis, Metering, Protection)

For example:

```
T:BWV;
```

The start register and the values to be written follow exactly the same rules and syntax as the definition for the Fixed Block Write, however, these are specified at the time the write operation is performed. For example, specifying "`S:100,1,2,3,4,-5`" as the write value for the tag "`T:BWV;`" would write values 1,2,3,4, and -5 to the registers 100, 101, 102, 103, and 104.

© 2011 Schneider Electric All Rights Reserved

# The Profile Wizard

This chapter discusses how to use the Profile Wizard to add an individual device profile to, or remove it from, a PowerLogic SCADA project. From the Profile Editor, you export profile data into a file that is available to be imported into a PowerLogic SCADA project. From there, use the Profile Wizard to add the device profile into the PowerLogic SCADA project. Real-time tags, control tags, and alarm tags in the Profile Editor become variable tags in PowerLogic SCADA. Alarm tags will also have a second tag in time-stamped digital alarms or time-stamped analog alarms, depending on whether they are on-board or PC-based.

To add a small number of profiles to the PowerLogic SCADA project, it makes sense to use the Profile Wizard. However, when you want to add a very large number of profiles, use the Automation Interface to save time. This feature allows you to add multiple devices in a single step. For information, see *Adding Multiple Devices through the Automation Interface* on page 107 for information.

Throughout the Profile Wizard, there are fields that will only accept a valid entry. They are marked with a red asterisk (!). The asterisk remains there until you enter a response that is of the correct length or includes only the acceptable characters. The asterisk disappears when you have entered a valid response.

In this section, you will find these topics:




© 2011 Schneider Electric All Rights Reserved

# Before You Use the Profile Wizard

Before you can add device profiles, you need to establish the folder structure and network addresses in the PowerLogic SCADA project: Launch the PowerLogic SCADA software: click Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Explorer. Three windows display:

- Citect Project Editor
- Citect Explorer
- Citect Graphics Builder

Have a copy of each device's connection information: communications protocol and IP address. You will enter this information when you add the devices.

In Citect Explorer, add a project that has the same name as the one set up in the Profile Editor (File > New Project). Then add the following configuration records in the Project Editor (Servers):

- at least one **cluster**; A cluster is a group of alarms servers, trends servers, reports servers, and I/O servers. For additional information about clusters, see Clustering in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).
- **a network server address for each cluster**: The name added here is linked to the IP address. This is the address to be used by all of the servers in the cluster. When using the host computer as the server, you can use the IP loopback address: 127.0.0.1.
- at least one of each relevant type of **server**: There are four types of server: alarm, report, trend, and IO.

  NOTE: When naming these servers, do not use the following names: IO_Server, Report_Server, Alarm_Server, Trend_Server, and Client. These names are used in Citect. Adding them here will cause a compilation error.

  An **alarm server** monitors all alarms and displays an alarm on the appropriate client(s) when an alarm condition becomes active.

  A **reports server** controls report processing; you can request reports at any time or when specific events occur.

  A **trends server** controls the accumulation and logging of trend information; this information provides a current and historical view of the system, and can be processed for display on a graphics page or printed in a report.

  An**I/O server** is a dedicated communications server that exchanges data between I/O devices and PowerLogic SCADA Control Clients.
- a **project** to PowerLogic SCADA: Then compile and correct any errors. After the first compile, run the Computer Setup Wizard (Project Editor > Tools > Computer Setup Wizard. This wizard prepares the computer for use with PowerLogic SCADA.

  NOTE: When running the Computer Setup Wizard, do not change the startup page (leave the ini settings as <default>). Changing the startup page at this point will result in navigation problems. To change the startup page, see *Setting a New Page as the Project Startup Page* on page 118.

## Adding the Cluster, Network Address, and Servers

Follow these steps to add the network address, which consists of at least one cluster, servers, and the related IP address. Each project must have at least one cluster, a network address, and one of each type of server that will be used. Clusters are used to logically and/or physically separate the servers in a system. Having multiple clusters on a large project (called scaleability) will help maintain system performance.

1. From the Citect Explorer screen, display the project: In the upper left corner of the screen, choose the project from the drop-down menu.

© 2011 Schneider Electric All Rights Reserved

2. From the Project Editor screen, click Servers > Clusters.
3. At the Cluster screen, type the cluster name.

   For the cluster to be able to communicate with system devices, its name must be between two and 16 characters, no spaces. It must be unique to the project.

   If there is only one cluster, the system will always automatically choose it for a project. If there are multiple clusters, you will always have to select the one that you want.
4. If desired, type a description in the Comment field.

   This field can be used for translation purposes. See *Entering Copy for Translation* on page 96 for more information.
5. Click Add.

   The new cluster name displays in the window.
6. Close the Cluster window.

   If a cluster already exists, you can add a new one: with the existing name displayed, click Replace to create a duplicate of the existing one; overwrite the name of the one you just added; finally click Add to save the name that you just typed. Both clusters will be in the system. To view all of the clusters added to this project, use the scroll bar on right side of the window.
7. Still on the Citect Explorer screen, click Servers > Network Addresses.

   Adding network servers enables you to handle data from large installations, and to use multiple servers.
8. At the Network Addresses screen, type a name for the network server; then enter the IP address for this server. This is the address that all of the servers in this project will use. If the servers are on the local machine, you can use the loopback address (127.0.0.1).

   The Comment field can be used for translation purposes. See *Entering Copy for Translation* on page 96 for more information.
9. Click Add, then exit the screen to save the server name.
10. Note the name so that you can re-use it for the remaining servers.
11. To add an alarm server, click Servers > Alarm Servers.
12. At the Alarm Servers screen, follow these steps:
    - At the Cluster Name screen, choose the cluster you will use for this server.
    - At the Network Addresses field, choose the network name that you added for this system.
    - In the Mode field, you can specify whether this server is primary or standby. The standby server will act as a backup, providing redundancy in case the primary server goes down.
    - In the Port field, select whether this is a primary or standby server (default: primary). You can have one primary and one standby server for each cluster.

      An additional set of prompts displays when you “expand” the screen. To expand, click F2 when the Alarm Servers screen is active. In the Publish Alarm Properties, choose whether alarm properties will be viewable as variable tags (default = false). If you set it to true, the alarm server monitors the port you specify as if it were an I/O connector.
    - Click add to save the new alarm server. Exit the window.
13. Repeat steps 11 and 12 for all other server types that you need to add.

    NOTE: When you delete a cluster, you need to also click File > Pack to completely delete it from the system.
14. Click File > Compile.
15. If there are errors and/or warnings after the page is compiled, do the following:


© 2011 Schneider Electric All Rights Reserved

a. At each error, click GoTo, which opens the location where the error has occurred.
b. Using the information in the error message, correct the error.
c. After all errors are addressed, re-compile to verify that the errors have been removed.

For additional information, click Help at the error screen.

## When Setting Up More than two I/O Servers per Cluster

If you need to add more than two I/O servers to a cluster, you need to define a redundant I/O device called *NetworkTagsDev* for each of the servers. If you do not do this, you can lose device status information during runtime.

If the cluster includes only one or two I/O servers, the I/O devices are automatically added when you add the cluster during Profile Wizard configuration (see *Adding the Cluster, Network Address, and Servers* on page 93). If a system has more than two I/O servers in a cluster, you must manually add the *NetworkTagsDev* I/O device for the remaining servers (after the first pair).

To create the board, port, and *NetworkTagsDev* I/O device, ensure the following in PowerLogic SCADA:

- all redundant *NetworkTagsDev I/O* devices have the same number
- the Startup Mode field is set to Standby; do this for all standby *NetworkTagsDev* I/O devices, including the one created by the Profile Wizard
- the Equipment field is set to <Cluster>_NetworkTagsDev

The field values for the forms in each of the I/O servers should be:

**Boards Form**

Board Name: <any unique name> (suggestion: BOARDy_SVRz)

Board Type: DISKXML

Address: 0

Leave everything else blank.

**Ports Form**

Port Name: <any unique name> (suggestion: PORTx_BOARDy_SVRz)

Port Number: <any unique number within the I/O server> (suggestion: x)

Board Name: <use the board name defined above>

Leave everything else blank.

**I/O Devices Form**

Name: NetworkTagsDev

Number: <same number as the one defined in the corresponding device>

Address: NetworkTagsDev

Protocol: DISKXML

Port Name: <use the port name defined above>

Startup Mode: Standby

Equipment: <Cluster> _NetworkTagsDev

Leave everything else blank.

**Notes**:

- Startup Mode is only visible when in extended form mode (press F2 to toggle between simple form mode and extended form mode, while in the I/O device form).

© 2011 Schneider Electric All Rights Reserved

- The Equipment field is hidden by default. To change it to visible, open units.dbf (in the project folder) in Excel.
- If the system has one or two I/O servers per cluster, the startup mode of the standby *NetworkTagsDev* I/O device could be set to StandbyWrite in the Profile Wizard. If the system has more than two I/O servers per cluster, the startup mode of all standby *NetworkTagsDev* I/O devices must be set to Standby.

  One side effect of this is that, when the system "fails over to a redundant I/O server," affected devices will momentarily be in communication failure as the system transitions to the redundant server.
- If the primary and redundant alarms servers are synchronizing, data will be slow to display in the Alarm Log and Events Log.

## Entering Copy for Translation

There are several “description” or “comment” fields throughout this tool. These fields can be used to create copy for translation purposes. If you type a Comment in this format:

@(XXX), where XXX = the copy that is to be translated,

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[your project](for Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[your project]). English.dbf contains terms that will be translated from English.

To create another language file for translation, set the Citect.ini parameter [Language]LocalLanguage to the specified language, then re-compile. So, for example, if you set this parameter to French, a French.dbf file is created in the project folder when you compile. You can then enter the translated text in the LOCAL field of the file. Repeat this same step for each additional language file you want in this project.

After translation, there will be an additional .dbf file for each translated language. At runtime, the user will be able to choose the .dbf file that is to be used in the display, thus changing the language used in the display.


© 2011 Schneider Electric All Rights Reserved

# Using the Wizard to Add and Remove Devices

This section describes how to use the Profile Wizard to manage device information in a PowerLogic SCADA project. Before you begin adding devices, make sure you have completed the tasks described in *Before You Use the Profile Wizard* on page 93.

You will use these procedures to add remove, or edit device data:

*Adding a Device to a PowerLogic SCADA Project* on page 97

*Adding a TCP Device to a PowerLogic SCADA Project* on page 97

*Adding a Serial Device to a PowerLogic SCADA Project* on page 99

*Adding a DNP3_TCP Device* on page 101

*Adding an IEC61850 Device to a PowerLogic SCADA Project* on page 103

*Naming Ports* on page 105

*Removing a Device From the PowerLogic SCADA Project* on page 106

*Editing Devices in a PowerLogic SCADA Project* on page 105

*Adding Multiple Devices through the Automation Interface* on page 107

## Adding a Device to a PowerLogic SCADA Project

The Profile Wizard allows you to add one device at a time. To add more than one device at a time, you can use the Automation Interface. See *Adding Multiple Devices through the Automation Interface* on page 107 for instructions.

Click one of the following links to display instructions for each type of protocol:

*Adding a TCP Device to a PowerLogic SCADA Project* on page 97

*Adding a Serial Device to a PowerLogic SCADA Project* on page 99

*Adding a DNP3_TCP Device* on page 101

*Adding an IEC61850 Device to a PowerLogic SCADA Project* on page 103

## Adding a TCP Device to a PowerLogic SCADA Project

Use this procedure when you want to add a generic TCP or Modbus TCP device to a PowerLogic SCADA project. Note that these instructions assume that you have two I/O servers, and that you will be renaming ports.

Before you begin, make sure that you have added at least one cluster and the appropriate servers for this project (see *Adding the Cluster, Network Address, and Servers* on page 93).

Then, make sure that you have set up each of these clusters in the Profile Wizard:

Cluster Setup: Before you add devices, you need to set up each cluster that was created for the project (you must have added at least one cluster to PowerLogic SCADA before you do this):

1. From the Profile Wizard, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.

© 2011 Schneider Electric All Rights Reserved

4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see *When Setting Up More than two I/O Servers per Cluster*).
5. At the Ready to perform action screen, click Next.

   If you have more than one cluster to add, repeat steps 3a through 3e for each cluster.

   When you are finished adding clusters and I/O servers, you return to the Profile Wizard welcome screen.

### Adding the TCP Device

Follow these steps to add TCP devices:

1. From the Citect Explorer screen, display the project to which you want to add the devices: In the upper left corner of the screen, choose the project from the drop-down menu.
2. From the Citect Project Editor, click Tools > Profile Wizard.

   The Profile Wizard welcome screen displays.
3. Click Create an I/O Device in the project, then click Next.
4. At the Choose profile screen, select the first device profile that you want to use to add a device to the PowerLogic SCADA project. Click Next.
5. At the Enter instance information screen, type a descriptive profile name, for example: *CM4Bay1Circuit1* (no spaces or punctuation; to allow space in PowerLogic SCADA, the preferred limit is 16 characters). The Comment field is stored in the equipment.dbf file.

   NOTE: There are several “description” or “comment” fields throughout the PowerLogic SCADA product. These fields can be used to create copy for translation purposes. If you type a Comment in this format:

   @(XXX), where XXX = the copy that is to be translated,

   the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[your project](for Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[your project]). English.dbf contains terms that will be translated from English.

   To create another language file for translation, set the Citect.ini parameter [Language]LocalLanguage to the specified language, then re-compile. So, for example, if you set this parameter to French, a French.dbf file is created in the project folder when you compile. You can then enter the translated text in the LOCAL field of the file. Repeat this same step for each additional language file you want in this project.

   At runtime, the user will be able to choose the .dbf file that is to be used in the display, thus changing the language used in the display.

   Click Next.
6. At the Select I/O servers screen, choose the primary and standby servers. You can only set the standby server if you check the "Supports Redundancy" box. Click Next.
7. At the Configure Sub-Profile Communications Method screen, choose the communications method used for the first sub-profile in this project. Click Next.


© 2011 Schneider Electric All Rights Reserved

8. At the Communications Settings screen, type the gateway address and station address for each of the servers. If you check the "Same as Primary" box for standby, you will use the same addresses for the primary and standby. Click Next.
9. At the Port Settings screen, you can rename each of the ports. A new port will be generated for each new name. Click Next. (See more information about naming ports in *Naming Ports* on page 105.)
10. Repeat steps 6 through 8 for each of the sub-profiles in this project.
11. At the Ready to perform action screen, click Next.

    After the devices are added, a screen displays telling you that the project was updated successfully.

    To view a detailed list of all of the device profiles and all operations performed in the project, check the View audit log box. The list displays after the device is added.

    To continue adding or removing device profiles, click Next. Repeat steps 3 through 8.
12. When you have finished adding devices, uncheck the Add/remove more equipment option, then click Finish.

    If you checked the View audit ... box, the list displays.

    If you did not check the Add/remove... box, the Profile Wizard closes. If you checked the Add/remove ... box, the Welcome screen displays again.
13. From any of the three PowerLogic SCADA screens, compile the project.
14. When all errors are corrected, click File > Run to view the runtime environment.

## Adding a Serial Device to a PowerLogic SCADA Project

Use this procedure when you want to add a generic serial or DNP3 via serial device to a PowerLogic SCADA project. Note that these instructions assume that you have two I/O servers, and that you will be renaming ports.

Before you begin, make sure that you have added at least one cluster and the appropriate servers for this project (see *Adding the Cluster, Network Address, and Servers* on page 93).

Then, make sure that you have set up each of these clusters in the Profile Wizard:

Cluster Setup: Before you add devices, you need to set up each cluster that was created for the project (you must have added at least one cluster to PowerLogic SCADA before you do this):

1. From the Profile Wizard, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see *When Setting Up More than two I/O Servers per Cluster*).
5. At the Ready to perform action screen, click Next.

   If you have more than one cluster to add, repeat steps 3a through 3e for each cluster.

   When you are finished adding clusters and I/O servers, you return to the Profile Wizard welcome screen.

© 2011 Schneider Electric All Rights Reserved

## Adding the Serial Device

Follow these steps to add serial devices.

1. From the Citect Explorer screen, display the project to which you want to add the device: In the upper left corner of the screen, choose the project from the drop-down menu.
2. From the Citect Project Editor, click Tools > Profile Wizard.

   The Profile Wizard welcome screen displays.

   Click Create an I/O Device in the project, then click Next.
3. At the Choose profile screen, select the first device profile that you want to use to add a device to the PowerLogic SCADA project. Click Next.
4. At the Enter instance information screen, type a descriptive profile name, for example: *CM4Bay1Circuit1* (no spaces or punctuation; to allow space in PowerLogic SCADA, the preferred limit is 16 characters). The Comment field is stored in the equipment.dbf file.

   NOTE: There are several “description” or “comment” fields throughout the PowerLogic SCADA product. These fields can be used to create copy for translation purposes. If you type a Comment in this format:

   @(XXX), where XXX = the copy that is to be translated,

   the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[your project](for Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[your project]). English.dbf contains terms that will be translated from English.

   To create another language file for translation, set the Citect.ini parameter [Language]LocalLanguage to the specified language, then re-compile. So, for example, if you set this parameter to French, a French.dbf file is created in the project folder when you compile. You can then enter the translated text in the LOCAL field of the file. Repeat this same step for each additional language file you want in this project.

   At runtime, the user will be able to choose the .dbf file that is to be used in the display, thus changing the language used in the display.

   Click Next.
5. At the Select I/O servers screen, choose the primary and standby servers. You can add information for the standby server if you check the "Supports Redundancy" box. Click Next.
6. At the Configure Sub-Profile Communications Method screen, choose the communications method used for the first sub-profile in this project. Click Next.
7. At the Communications Settings screen, type all of the information for each server (com port, baud rate, etc.). If you check the "Same as Primary" box for standby, you will use the same addresses for the primary and standby. Click Next.
8. At the Port Settings screen, you can rename each of the ports. (See more information about naming ports in *Naming Ports* on page 105.)
9. Repeat steps 6 through 8 for the remaining sub-profiles.
10. When you finish adding the last sub-profile, the Ready to perform action screen displays Click Next.

    After the devices are added, a screen displays telling you that the project was updated successfully.


© 2011 Schneider Electric All Rights Reserved

To view a detailed list of all of the device profiles and all operations performed in the project, check the View audit log box. The list displays after the device is added.

To continue adding or removing device profiles, click Next. Repeat steps 3 through 8.

11. When you have finished adding devices, click Finish at the Project updated successfully screen.

    If you checked the View audit ... box, the list displays.

    The Profile Wizard closes.

12. From any of the three PowerLogic SCADA screens, compile the project.
13. When all errors are corrected, click File > Run to view the runtime environment.

## Adding a DNP3_TCP Device

Use this procedure when you want to add a DNP3_TCP device to a PowerLogic SCADA project. Note that these instructions assume that you have two I/O servers, and that you will be renaming ports.

Before you begin, make sure that you have added at least one cluster and the appropriate servers for this project (see *Adding the Cluster, Network Address, and Servers* on page 93).

Then, make sure that you have set up each of these clusters in the Profile Wizard:

Cluster Setup: Before you add devices, you need to set up each cluster that was created for the project (you must have added at least one cluster to PowerLogic SCADA before you do this):

1. From the Profile Wizard, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see *When Setting Up More than two I/O Servers per Cluster*).
5. At the Ready to perform action screen, click Next.

   If you have more than one cluster to add, repeat steps 3a through 3e for each cluster.

   When you are finished adding clusters and I/O servers, you return to the Profile Wizard welcome screen.

### Adding the DNP3_TCP Device

Follow these steps to add TCP devices:

1. From the Citect Explorer screen, display the project to which you want to add the devices: In the upper left corner of the screen, choose the project from the drop-down menu.
2. From the Citect Project Editor, click Tools > Profile Wizard.

   The Profile Wizard welcome screen displays.

   Click Create an I/O Device in the project, then click Next.
3. At the Choose profile screen, select the first device profile that you want to use to add a device to the PowerLogic SCADA project. Click Next.

© 2011 Schneider Electric All Rights Reserved

4. At the Enter instance information screen, type a descriptive profile name, for example: *CM4Bay1Circuit1* (no spaces or punctuation; to allow space in PowerLogic SCADA, the preferred limit is 16 characters). The Comment field is stored in the equipment.dbf file.

   NOTE: There are several “description” or “comment” fields throughout the PowerLogic SCADA product. These fields can be used to create copy for translation purposes. If you type a Comment in this format:

   @(XXX), where XXX = the copy that is to be translated,

   the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[your project](for Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[your project]). English.dbf contains terms that will be translated from English.

   To create another language file for translation, set the Citect.ini parameter [Language]LocalLanguage to the specified language, then re-compile. So, for example, if you set this parameter to French, a French.dbf file is created in the project folder when you compile. You can then enter the translated text in the LOCAL field of the file. Repeat this same step for each additional language file you want in this project.

   At runtime, the user will be able to choose the .dbf file that is to be used in the display, thus changing the language used in the display.

   Click Next.
5. At the Select I/O servers screen, choose the primary and standby servers. You can only set the standby server if you check the "Supports Redundancy" box. Click Next.
6. At the Configure Sub-Profile Communications Method screen, choose the communications method used for the first sub-profile in this project. Click Next.
7. At the Communications Settings screen, type the IP address, port number, and device address for each of the servers.

   NOTE: The DNP3 port number is by definition 20000. You must type 20000 here in order for communications to work correctly.

   If you check the "Same as Primary" box for standby, you will use the same addresses for the primary and standby. Click Next.
8. At the Port Settings screen, you can rename each of the ports. A new port will be generated for each new name. Click Next. (See more information about naming ports in *Naming Ports* on page 105.)
9. Repeat steps 6 through 8 for each of the sub-profiles in this project.
10. At the Ready to perform action screen, click Next.

    After the devices are added, a screen displays telling you that the project was updated successfully.

    To view a detailed list of all of the device profiles and all operations performed in the project, check the View audit log box. The list displays after the device is added.

    To continue adding or removing device profiles, click Next. Repeat steps 3 through 8.
11. When you have finished adding devices, uncheck the Add/remove more equipment option, then click Finish.

    If you checked the View audit ... box, the list displays.

    If you did not check the Add/remove... box, the Profile Wizard closes. If you checked the Add/remove ... box, the Welcome screen displays again.


© 2011 Schneider Electric All Rights Reserved

12. From any of the three PowerLogic SCADA screens, compile the project.
13. When all errors are corrected, click File > Run to view the runtime environment.

## Adding an IEC61850 Device to a PowerLogic SCADA Project

Use this procedure when you want to add an IEC61850 device to a PowerLogic SCADA project. Note that these instructions assume that you have two I/O servers, and that you will be renaming ports.

There will be an additional .dbf file for each translated language. At runtime, the user will be able to choose the .dbf file that they want to use in the display. This changes the language in the display.

Before you begin, make sure that you have added at least one cluster and the appropriate servers for this project (see *Adding the Cluster, Network Address, and Servers* on page 93.

Then, make sure that you have set up each of these clusters in the Profile Wizard:

Cluster Setup: Before you add devices, you need to set up each cluster that was created for the project (you must have added at least one cluster to PowerLogic SCADA before you do this):

1. From the Profile Wizard, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see *When Setting Up More than two I/O Servers per Cluster* on page 95).
5. At the Ready to perform action screen, click Next.

   If you have more than one cluster to add, repeat steps 3a through 3e for each cluster.

   When you are finished adding clusters and I/O servers, you return to the Profile Wizard welcome screen.

### Adding the IEC 61850 Device

1. From the Citect Explorer screen, display the project to which you want to add the devices: In the upper left corner of the screen, choose the project from the drop-down menu.
2. From the Citect Project Editor, click Tools > Profile Wizard.

   The Profile Wizard welcome screen displays.

   Click Create an I/O Device in the project, then click Next.
3. At the Choose profile screen, select the first device profile that you want to use to add a device to the PowerLogic SCADA project. Click Next.
4. At the Enter instance information screen, type a descriptive profile name, for example: *Bay1Circuit1* (no spaces or punctuation; to allow space in PowerLogic SCADA, the preferred limit is 16 characters). The Comment field is stored in the equipment.dbf file.

   NOTE: There are several “description” or “comment” fields throughout the PowerLogic SCADA product. These fields can be used to create copy for translation purposes. If you type a Comment in this format:

© 2011 Schneider Electric All Rights Reserved

@(XXX), where XXX = the copy that is to be translated,

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[your project](for Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[your project]). English.dbf contains terms that will be translated from English.

To create another language file for translation, set the Citect.ini parameter [Language]LocalLanguage to the specified language, then re-compile. So, for example, if you set this parameter to French, a French.dbf file is created in the project folder when you compile. You can then enter the translated text in the LOCAL field of the file. Repeat this same step for each additional language file you want in this project.

At runtime, the user will be able to choose the .dbf file that is to be used in the display, thus changing the language used in the display.

### LDName

In the Additional Information section at the bottom, you can change the original logical device names for the IED. This is required only if the logical device name was changed in the SCL file that was imported into the Profile Editor.

### BRCBs and URCBs

In the Additional Information, you can also enter BRCB or URCB information. BRCBs (buffered report control blocks) and URCBs (unbuffered report control blocks) can be used to return data in blocks rather than in individual tags. To enter either one, you need to have downloaded an SCL file for the device in question. When you click the line to add data, you must browse to the SCL file and select the BRCB/URCB you want. You will need the logical device, logical node, and RCB names. The Help column gives examples of the formatting that is required.

Click Next.

5. At the Communications Settings screen, browse to the location where you have saved the SCL file. If there is only one IED, it displays automatically; otherwise, choose the correct device. Click Next.
6. At the Ready to perform action screen, click Next.

   After the devices are added, a screen displays telling you that the project was updated successfully.

   To view a detailed list of all of the device profiles and all operations performed in the project, check the View audit log box. The list displays after the device is added.

   To continue adding or removing device profiles, click Next. Repeat steps 3 through 6.
7. When you have finished adding devices, uncheck the Add/remove more equipment option, then click Finish.

   If you checked the View audit ... box, the list displays.

   If you did not check the Add/remove... box, the Profile Wizard closes. If you checked the Add/remove ... box, the Welcome screen displays again.
8. From any of the three PowerLogic SCADA screens, compile the project.
9. When all errors are corrected, click File > Run to view the runtime environment.


© 2011 Schneider Electric All Rights Reserved

## Naming Ports

The Profile Wizard does not take into account that multiple projects might be 'linked together' via a global include project. For instance, it does not allow you to specify a unique port name and port number, such that they will not conflict with other projects.

There are three possibilities:

- Protocols that support port name changes: includes Generic TCP and MODBUS TCP
- Protocols that support re-use of ports only: see the table below for protocols and settings that need to match
- Protocols that do not support port name changes: all protocols not mentioned above

The table below shows the settings that must match between the protocols for that column. For example, if you combine two generic serial protocols or a generic serial with a DNP3 via serial, all of the checked items need to match between them.

| | Generic Serial, DNP3 via Serial | MODBUS RTU via Serial | DNP3 via TCP/IP, IEC 60870-5-104 via TCP/IP, MODBUS RTU via Gateway |
|---|---|---|---|
| Board Type | X | X | X |
| I/O Server Name | X | X | X |
| Port Number | X | X | |
| Baud Rate | X | X | |
| Data Bits | X | X | |
| Stop Bits | X | X | |
| Parity | X | X | |
| IP Address | | | X |
| Network Port Number | | | X |
| All attached I/O devices must use the same protocol. | | X | X |

Using the Port Settings page in the Profile Wizard, you can name ports. See *Adding a Device to a PowerLogic SCADA Project* on page 97 for more information.

## Editing Devices in a PowerLogic SCADA Project

### Editing a Profile and Adding it Back to the Project

This feature works only if the device was added in version 7.20. You cannot update devices that were added in version 7.10.

After you have added devices to the PowerLogic SCADA project, and you make changes to the device in the Profile Editor (e.g., add a large number of tags),you can use the Profile Wizard to bring the changes in the PowerLogic SCADA project.

NOTE: If you have made manual changes to the profile in PowerLogic SCADA (Citect), do not use this process: you could corrupt your data. You must delete the device from the PowerLogic SCADA project, re-export it from the Profile Editor, and add it back to Citect via the Profile Wizard.

To use this method of importing changes:

© 2011 Schneider Electric All Rights Reserved

1. Make the changes in the Profile Editor. Make sure you refresh the tags before you continue.
2. Export the project from the Set Up Projects tab.
3. From the Citect Project Editor, click Tools > Profile Wizard.
4. At the Welcome screen, check Update one or more I/O Device(s) in the project. Click Next.
5. At the choose update type screen, check whether you want to update all instances in a profile, or just one instance. Click Next.
6. Note the two possibilities:
   a. If you selected all instances, choose the profile, and click Next.
   b. If you selected "one instance," the Update profile instance screen displays. From the drop down list, choose the instance you want to update.
7. At the Ready to perform action screen, note the instance(s) you are about to update. If you want to change your choice, click Back.

   To compress the project files in PowerLogic SCADA, check "Pack databases after update."

   When you have made the update choice you want, click Next.

   When the update is finished, the Project updated successfully screen displays. You can view an audit log of changes that have been made, process more changes, or click Finish to leave the Profile Wizard.

### Editing a Device in PowerLogic SCADA Only

If, when you added the device to the PowerLogic SCADA project, you entered incorrect information (e.g. wrong IP address), do the following:

1. Delete the device from the PowerLogic SCADA project: Use the “Remove a device from the project” feature in the Profile Wizard.
2. In the Profile Wizard, add the device back to the PowerLogic SCADA project.

### Adding Device Data in PowerLogic SCADA Only

If you need to add a small amount of data to a device that is in the PowerLogic SCADA project (e.g., add a single tag), add it directly in PowerLogic SCADA. Be sure that you also add it to the device in the Profile Editor so that it is available for other devices in the future.

## Removing a Device From the PowerLogic SCADA Project

1. From the Citect Explorer screen, display the project from which you want to remove the device: In the upper left corner of the screen, choose the project from the drop-down menu.
2. From the Citect Project Editor, click Tools > Profile Wizard.

   The Profile Wizard welcome screen displays.
3. Click Remove a device from the project, then click Next.
4. At the Remove a device screen, click the dropdown menu to display the equipment names that were used when the device profiles were added in the Profile Wizard. From this list, select the device that you want to remove. Click Next.
5. At the Ready to perform action:

   To compress the project files after removing this profile, click Pack databases.

   Click Next.


© 2011 Schneider Electric All Rights Reserved

After the device is deleted, a screen displays telling you that the project was updated successfully.

To view a detailed list of devices that you have added or deleted, check the View audit log box. The list will show all of the device data that has been added, as well as the data that has been removed in this session. The list displays after you click Finish.

6. To continue removing devices, check Add/remove more devices, then click Next. Repeat steps 4 and 5.
7. When you have finished removing devices, uncheck Add/remove more devices, then click Finish.

   If you checked the View audit ... box, the list displays.

   If you did not check the Add/remove... box, the Profile Wizard closes. If you checked the Add/remove ... box, the Welcome screen displays again.

## Adding Multiple Devices through the Automation Interface

The Automation Interface allows you to add multiple devices at a time, instead of using the Profile Wizard, which only adds one device at a time. Follow these instructions to add large quantities of devices into the PowerLogic SCADA project. (This example is for MODBUS TCP. See the table at the end of these instructions for other comms methods.)

NOTE: If you install PowerLogic SCADA on a computer that does not have Microsoft Excel installed, you will not be able to run the Automation Interface. To ensure proper Microsoft COM registration with Excel, you must install Excel before you install PowerLogic SCADA. If Excel is installed after PowerLogic SCADA, you will see an "out of memory" message when you try to run the macro. The solution is to uninstall both PowerLogic SCADA and Excel. After that, first reinstall Excel, then reinstall PowerLogic SCADA.

1. For an existing project: Before you begin, make a backup copy of your project. See *PowerLogic SCADA Backup* on page 129 for instructions.

   For a new project: In the Citect Explorer, add a new project. From the Project Editor, define a cluster; and add alarm, trend, and I/O servers (see *Before You Use the Profile Wizard* on page 93).
2. In the Profile Editor, create a project that includes the devices included in this installation. Give the project the same name you used in step 1.
3. Open Excel 2003 or Excel 2007 and enable macros (Click the Developer tab, then Macro security. Click Enable all Macros).
4. In Excel, open *Example Automation.xls* (Program Files\Schneider Electric\PowerLogic SCADA 7.20\Bin). This sample spreadsheet can be edited to include the communications methods you will use. Then, enter the devices that you want to add to the PowerLogic SCADA project in the spreadsheet.
5. From the Setup worksheet at the bottom of the spreadsheet, make these changes:

   a. Cell B2: Enter the path to Citect.ini:

      For Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\Config.

      Vista operating systems: ProgramDataSchneider Electric\PowerLogic SCADA 7.20\Config.

   b. Cell B3: Type the project name.

   c. Cell B4: Enter the path to the location of *equipment.profiles:*

      For Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\ProjectName).

© 2011 Schneider Electric All Rights Reserved

For Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\ProjectName).

6. From the Input worksheet at the bottom of the spreadsheet, enter the following information for each device that you want to add:
   a. Profile Name: Type the complete name (to view exact addresses, open *equipment.profiles* in Notepad), e.g.: SchneiderElectric.Protection Device.Sepam S42 Full
   b. Name: Enter the device name (no spaces or punctuation, limit of 32 characters). This becomes the equipment name in the PowerLogic SCADA project. You will use this when adding genies to drawings.
   c. Cluster: Type the cluster name from step 1.
   d. Primary IO Server Name: Type the I/O server added in step 1. It becomes the primary server.
   e. CommsMethod: Type the communications protocol being used, e.g., MODBUS/RTU via Gateway. See table below for alternate communication connections.
   f. Additional Prompts will display, depending on the communication connection being used. See the table below for a list of data required for each comms method.
   g. EquipmentAddress: Type the device address (required only for MODBUS/RTU and MODBUS/RTU via Gateway; optional for other comms methods).
7. Close Citect Explorer if it is open.
8. In Example Automation.xls, click Tools > Macro > Macros. Highlight the macro called *ThisWorkbook.AddDevices*, then click Run.

   Devices are created, according to the information you added. When device creation is complete, a message displays.

   NOTE: For large systems, this can take quite a long time. While the macro is running, there is no feedback on the screen.
9. Launch PowerLogic SCADA and verify communication for all of the devices listed in the spreadsheet.

If *Example Automation.xls* does not include a communications connection that you want to use, you can add the new connection. The following table lists connections and the required information for each.

| CommsMethod | ComPort | Baud Rate | DataBits | StopBits | Parity | IP Address | Equipment Address | |
|---|---|---|---|---|---|---|---|---|
| Generic Serial [1] | X | X | X | X | X | | X | 2 |
| MODBUS/RTU | X | X | X | X | X | | X | |
| Generic TCP | | | | | | X | X | 2 |
| MODBUS/RTU via Gateway | | | | | | X | X | |
| MODBUS/TCP | | | | | | X | X | 2 |

1 Generic Serial comms must be set up via Gateway in this release.

2 Optional


© 2011 Schneider Electric All Rights Reserved

# Using the OLE Automation Interface to Add and Remove Devices

In this section, you will find these topics:



There is also an OLE Automation interface that allows you to add and remove devices from a project. The interface can be used by VB, .NET, and by C++ clients. It is the building block for building a bulk interface based on Excel.

**IEquipmentAutomation**. The equipment automation interface facilitates reading equipment profiles, selecting a profile by name, setting the required information and saving the profile to a project.

**INameValueCollection**. The **INameValueCollection** interface provides the means to set the attributes and comms properties. The interface provides the method **SetValue** for setting a named value for the currently selected profile. If the name is not recognised, the method will return false and errors will be handled by the current error output method. In the case where an error has occurred, the **GetErrorMessages** may be called on the **EquipmentAutomation** interface to get more information about what went wrong.

## Examples

### C#

After a C# project is created, a reference to the EquipmentAutomation.dll should be established.

Using the equipment automation will be made easier by using the correct namespace as shown:

```
using Citect.CitectSCADA.Equipment.Automation;
```

The following is an example of some typical C# code that will select a profile, set it up and save it.

```
EquipmentAutomation myAuto = new EquipmentAutomation();
myAuto.IniFile =
@"C:\Documents and Settings\All Users\Application Data\Schneider
Electric\PowerLogic SCADA 7.20\Config\citect.ini";

myAuto.ReadProfiles(
@"C:\Documents and Settings\All Users\Application Data\Schneider
Electric\PowerLogic SCADA 7.20\User\TestProfiles\equipment.profiles");

string profileName = "Schneider Electric.Protection Device.Branch Circuit Monitor Full";

if (myAuto.SelectProfile(profileName))
{
    If (myAuto.SelectProfile(profileName))
    {
        NameValueCollection myAttributes = myAuto.ProfileAttributes;
        myAttributes.SetValue("NAME", "yourEquipName");
        myAttributes.SetValue("CLUSTER", "yourCluster");
        myAuto.ServerNamePrimary = "PrimaryServer";
        myAuto.SetCommsMethod("Modbus/RTU via Gateway");
```

© 2011 Schneider Electric All Rights Reserved

```
        INameValueCollection myCommsProperties =
        myAuto.CommsMethodPropertiesPrimary;
        myCommsProperties.SetValue("IPAddress", "10.0.0.1");
        myCommsProperties.SetValue("EquipmentAddress", "1");
        equipmentAutomation.SaveProfile("TestProfiles");
    }
}
```

### Excel VBA

An example spreadsheet, called “Example Automation.xls,” is provided in the bin directory. This spreadsheet has a “Setup” and an “Input” sheet. The Setup sheet contains information about the ini file location, the project name, and the profile that is used. The Input page contains information that is used by the macro to select a profile, set the attributes, set the IO server name, select the comms method and set the appropriate comms properties.

## Command Line Interface

The Device Wizard allows the wizard to be launched from the command line. It uses the flag /i to specify the location of the Power SCADA Ini file (usually found in the config directory of the user/data directory), and the flag /p to specify the project name. The arguments to these flags must immediately follow the flag without a space. It is recommended to use quotes around the ini filename.

In addition to launching the wizard, the Device Wizard allows a merge of a device profile XML file with the current PowerLogic SCADA “master” device profile XML file. The given file must conform to the profiles schema.

Additional help may be obtained by using the /h flag.

EquipmentWizard.exe **/?**

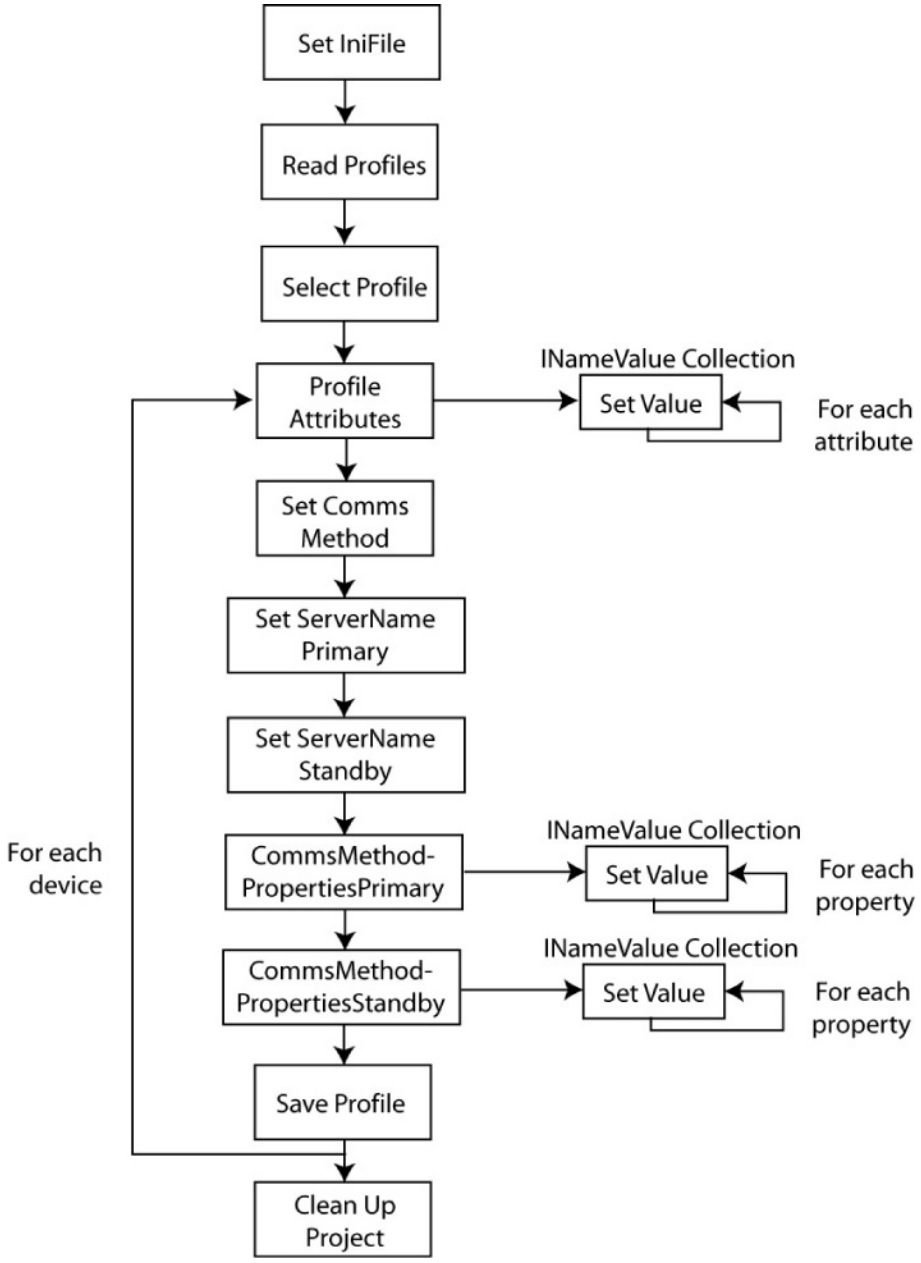
EquipmentWizard.exe **/p**”ProjectName” **/i**”iniFile”

EquipmentWizard.exe **/p**”ProjectName” **/i**”iniFile” **/m**”MergeProfileName”

## Interface Workflow

The general workflow, when adding devices via the automation interface, is illustrated in the following flowchart. Success values can be checked at each step in the process by checking the Boolean return value or by checking the engine errors.


© 2011 Schneider Electric All Rights Reserved




### Prerequisite information

Before creating a profile in a project, key information about the profile and the project in which it is being added should be established. This information can be established by examining the profile definition in the equipment.profiles file for the project. Attribute names and the supported comms methods must be known in order to know what needs to be set. The cluster and the IO server names from the project that the equipment is being added to should also be established.

Error messages can also provide useful feedback, and a way of discovery for what is required. The profile will not be saved until all required information is determined to be valid and complete.

© 2011 Schneider Electric All Rights Reserved

### Interface

The automation interface is designed to lead the developer through the workflow, providing useful feedback along the way in the form of error messages as to what was missing or incorrect. The interface is a COM automation interface that allows multiple profiles to be saved to a project.

**IniFile [Property] —** The equipment engine relies on the location of the ini file to determine where projects are stored and for validating that a profile can be successfully imported into the selected project.

**ReadProfiles** —The equipment engine reads the given profile file into memory and ensures that it is matches the expected schema.

**SelectProfile** — Selecting a profile initialises the engine for a specific profile. A profile is selected by referring to the qualified name. The qualified name is made up of the categories that contain the profile separated by a '.'.

**ProfileAttributes [Property]** — Attributes are set by first retrieving an **INameValueCollection** from the **ProfileAttributes** property. The **INameValueCollection** provides an interface method to set a named value. This will set the attribute with the given name for the selected profile in the equipment engine. SetValue returns a Boolean value indicating success.

**ServerNamePrimary and ServerNameStandby [Properties]** — The name of the primary IO server is required to be set (ServerNamePrimary) before saving a profile. A standby server name may also be optionally specified (ServerNameStandby). The server names must be valid server names in the project that the profile is being imported into.

**SetCommsMethod** — Each profile specifies which comms methods it supports. Setting the comms method requires that the currently selected profile supports the comms method specified.

**CommsMethodPropertiesPrimary and CommsMethodPropertiesStandby[Properties]** — Comms properties are also set via **INameValueCollection** which is returned when calling **commsMethodPropertiesPrimary** or **commsMethodPropertiesStandby**. If a standby server was specified, the comms properties for the standby server are required to be set. The comms properties required depend on the comms method selected. Below is a table listing the comms properties required for each of the comms methods.

| CommsMethod | ComPort | Baud Rate | DataBits | StopBits | Parity | IP Address | Equipment Address | |
|---|---|---|---|---|---|---|---|---|
| Generic Serial[1] | X | X | X | X | X | | X | 2 |
| MODBUS/RTU | X | X | X | X | X | | X | |
| Generic TCP | | | | | | X | X | 2 |
| MODBUS/RTU via Gateway | | | | | | X | X | |
| MODBUS/TCP | | | | | | X | X | 2 |

1 Generic Serial comms must be set up via Gateway in this release.

2 Optional

**SaveProfile** — SaveProfile takes the name of the project to which the profile is being saved. Saving a profile involves writing all output values defined in the profile definition to the specified project. A two-phase commit process is used to ensure that data is


© 2011 Schneider Electric All Rights Reserved

valid before committing changes permanently. Saving profiles to a different project will result in an automatic cleanup of the first project. See CleanupProject for more information.

**ResetSelectedProfile** — A call to ResetSelectedProfile will clear data associated with the selected profile, as well as clearing any accumulated errors. After saving a profile successfully, this method will be called automatically to allow the next profile to be saved.

**DeleteEquipment** — DeleteEquipment requires the project name that the equipment is to be deleted from as well as the name of the equipment that is being deleted. It will scan through the project looking for records that were added for a specific piece of equipment and will remove them accordingly. Removal of file outputs which were defined in the profile from which the equipment was added is not performed as these files may contain sensitive data that has since been added to these files. Additionally, those dbf entries in the profile under dbf-file that do not have an equip field are not removed, as there is no way of determining if they are exclusively used by this piece of equipment.

**CleanupProject** — CleanupProject performs operations such as re-indexing of dbf files and packing dbf files. Errors detected by this phase are reported as warnings and do not trigger a rollback. This operation should be called after all save operations have been performed on a project.

**NoErrors [Property]** — This is a Boolean property that can be used to determine if any errors have been collected.

**GetErrorMessages** — Detailed error messages may be obtained by calling **GetErrorMessages**, which returns a string with errors and warnings on multiple lines.

**ErrorOutput [Property]** — The error handling approach be set with the **ErrorOutput** property. Error Handling.

The values used for the error output are:

- None = 1
- Dialog = 2 [default]
- Console (silent if run from excel) = 4

**Error Handling** — Each interface method returns a Boolean indicating the success of that operation. Accumulated errors will prevent progression through the workflow and as such, this return state should be examined at each stage. A call to **ResetSelectedProfile** will clear accumulated errors, but it is recommended that errors are addressed before you proceed.

Information about errors and warnings can be obtained through a call to **GetErrorMessages**. By default, a dialog will display with error information at each step of the workflow, but this can be controlled with the **ErrorOutput** property.

© 2011 Schneider Electric All Rights Reserved

# Graphics: Using the Design Time Environment

This chapter discusses how to use genies to create one-line diagrams for customers. In this section, you will find these topics:




© 2011 Schneider Electric All Rights Reserved

# Typical Workflow

When creating a graphics page, these are the steps you will usually follow:

1. Before you begin, make sure you have:
   a. created a project in the Profile Editor
   b. added a project with the same name to PowerLogic SCADA; added at least one cluster, network address, and server
   c. exported the project from the Profile Editor
   d. used the Profile Wizard to add devices to PowerLogic SCADA
2. From the Graphics Builder window, add a new page, using the style and page template you prefer. Instructions assume that you will use the default template; but if you create your own template, use the Menu Configuration tool to add menu headings that will display on the one-line in runtime. (See *Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121 for more information about changing the appearance of the graphics page.)
3. Add genies to the page to create the one-line drawing.
4. Compile and run the project. Note any errors and warnings; correct all errors. Review warnings for problems such as missing tags.
5. Run the project to view the graphics page in the runtime environment.

© 2011 Schneider Electric All Rights Reserved

# The PowerLogic SCADA Pages

When you launch PowerLogic SCADA (Start > Programs > Schneider Electric > PowerLogic SCADA Explorer), three windows are opened:

- **Citect Explorer**: Use this application to explore and add projects. PowerLogic SCADA is shipped with a project that has example page configuration. Project names are listed in the left-hand pane.
- **Citect Project Editor**: Use this application to edit the SCADA configuration. For example, use it to add clusters, network addresses, and servers; and to launch the Profile Wizard and the Project Color Change tool.
- **Citect Graphics Builder**: Use this application to build graphic display pages for a project. A standard PowerLogic library of genies are provided. The genies contain logic such as control outputs and rack-in/rack-out. Once assembled and opened in the runtime environment, they provide an interactive view of an installation's power system.


© 2011 Schneider Electric All Rights Reserved

# Creating a PowerLogic SCADA Project

The information in this section is based on the Normal template from the PLS_Style_1 library. The Normal template includes all of the tabs and buttons that are described in the topics here. If you want to edit selections on the tabs and buttons, use the Menu Configuration tool. For instructions on editing templates with the Menu Configuration tool, see *Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121.

When you modify the template or create a completely new template, the information provided here for creating graphics pages may only apply as a guide.

When you rename a project (using the .Project Properties option in Citect), you will see error messages when you try to add new projects. To fix this problem, just shut down and then restart PowerLogic SCADA.

In this section, you will find these topics:



## Reusing Projects Created in the Profile Editor

You may want to create a “standard” project that can be reused for different installations. For instructions on saving and then reusing projects, follow these steps:

1. Export the project from the Set Up Project tab of the Profile Editor.
2. Click the folder icon beside the Export link:

3. Copy the file (Equipment.profiles) that displays. If you need to use this file to another computer, you can move it to a portable drive.
4. On the PowerLogic SCADA server computer, paste Equipment.profiles in this location:

   Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[Project]

   Vista operating systems: [Drive Letter]:\ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[Project]

   where:

   [Drive Letter]: is the drive on which you installed the PowerLogic SCADA server

   the Application Data and Program Data folders are not hidden (set the folder view for “view hidden folders”)

   [Project] is the name of the project you are creating; you must have already added this project to PowerLogic SCADA (see Before You Export, above).
5. Be sure you have created the files described in *Before You Begin* on page 118.

© 2011 Schneider Electric All Rights Reserved

## Before You Begin

Before you start adding data in the PowerLogic SCADA project, make sure that you have:

- used the Profile Editor to add all of the device types, device profiles, and projects
- from Citect Explorer, created a project in PowerLogic SCADA; from the Citect Editor, added clusters, network addresses, and servers
- exported devices from the Profile Editor
- added devices into the PowerLogic SCADA project, using the Profile Wizard

## Adding a New Graphics Page

You need to add a project in two places before you add pages. First add the project to PowerLogic SCADA, from the Citect Explorer screen (see steps below). Then add it to the Profile Editor. See*Adding, Editing, or Deleting a Project* on page 66 for instructions.

1. From the Citect Explorer screen, choose the project for which you will create a graphic.
2. On the Graphics Builder screen, click File > New > Page.
3. On the Use Template popup, set the resolution to SXGA (or default), and uncheck the Title Bar option. Select the page template (for example, normal). Click OK.

   NOTE: If you use any style that is not preceded by “pls,” the PowerLogic SCADA graphics features will not be included. If you want to change the attributes of the default style, copy *pls_style_1* and paste it in your project; rename it, and make the desired changes. The new template can then be used for your pages.

   The graphics page displays in design-time mode.
4. To make changes to the menus and tabs that display on the screen, use Menu Configuration tool. See *Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121 for instructions on making these changes.
5. Save the graphics page, giving it the desired name and selecting the project that you have just added.

## Setting a New Page as the Project Startup Page

After you add a new page, follow these steps to make it be the system starting page. If you do not make these settings correctly, the system will not properly execute the startup routine, and the browser navigation buttons will not work.

1. Open the Parameters dialog (Project Editor > System > Parameters).
   a. In the Section Name field, type MultiMonitors.
   b. In the Name field, type StartupPage.
   c. In the Value field, type the name of the new page.
   d. If you intend to use multiple monitors, and want to use a different startup page for a particular monitor, you can specify it by adding another parameter with the name of StartupPage<n>, where <n> is the monitor number (for example, StartupPage2, for startup page on the second monitor).
2. In the Computer Setup Wizard (General Options Page), leave the ini settings as <default>.


© 2011 Schneider Electric All Rights Reserved

## Creating a One-Line on a Graphics Page

You will build a one-line by adding genies to the new page. For more information about genies, see Genies and Super Genies in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).

To begin creating a one-line, start adding genies to the page:

1. Click the “genie” icon on the toolbar:

2. From the Paste Genie screen, choose a library (to ensure that all PowerLogic SCADA features are available, choose a library that begins with “pls”). If you want to edit a genie or create a new one, see *Creating and Editing Genies* on page 123 for instructions.
3. From the available icons in the selected library, select a genie; click OK to paste the genie on the graphics page.
4. A popup window displays for you to enter genie properties. Enter the requested information (descriptions below for circuit breakers, switchgear, and busbars only):

   **Circuit breakers, both ANSI-type and IEC-type**

   NOTE: Type 1 genies are of two styles: NC (no control) and C (control). The NC genies have no open/close control buttons; the C genies have open/close buttons.

   NOTE: If you choose to resize a circuit breaker genie after you paste it into a page, you must keep the relative dimensions (proportions) the same. Otherwise, the racked in/racked out animation will not display correctly.

   a. Equipment: From the drop-down list, choose the name of equipment represented by the genie (this is an equipment name that you entered in the Profile Wizard or the Automation Interface, e.g., *CM4Main*). To view the equipment you can use for this page, you must have the project selected in the Citect Explorer window.
   b. Busbar Connections: Where you are connecting to busbars, type the numbers for the connection source and destination. These are arbitrary, and only indicate how the genies in the one-line fit together on the page. Busbar numbers can be from 0 to 74. Numbers must be unique within each page. We recommend that they be consistent across pages for future compatibility.
   c. Label (currently, labels are only available on Type 2 circuit breakers): Type 2 circuit breakers have two label fields, upper (incomer) and lower.Type the name you want to appear for each label. **Note**: If you type too many characters to the upper label, it will overlap the earth switch, and will not be readable.
   d. Value Type: From the drop-down list, select the tag you want to use for this genie. This information displays in the lower left corner of the genie in run time.

      The tag you choose here causes real-time data (for example, “currents”) to display on the runtime environment of the Graphics page. If you do not select a tag here, the status screen will have a blank in that spot. This tag also determines what displays in the popup (volt, amp, watts, etc.) See *Using the Equipment Pop-Up Page* on page 150 for more information.
   e. Units: From the drop-down list, select the unit (created in the Profile Editor). Units will display beneath the Value Type on the run time screen.
   f. Multiplier: Enter the multiplier you want to use for the units in this graphic.

   **Switchgear**

© 2011 Schneider Electric All Rights Reserved

a. Busbar Connections: Where you are connecting to busbars, type the numbers for the connection source and destination. These are arbitrary, and only indicate how the genies in the one-line fit together on the page. Busbar numbers can be from 1 to 100. Numbers must be unique within each page. We recommend that they be consistent across pages for future compatibility.

b. Label (currently, labels are only available on larger circuit breakers): Type the name you want to appear on upper left corner of the page in the runtime environment.

c. Equipment: From the drop-down list, choose the name of equipment represented by the genie (this is an equipment name that you entered in the Profile Wizard or the Automation Interface, e.g., *CM4Main*). To view the equipment you can use for this page, you must have the project selected in the Citect Explorer.

**Busbars**

a. Busbar Number: Use busbar numbers to associate devices within the drawing; they are also an aid to help you set up animation for genies in the drawing. The busbar number you enter here is also used on the associated device. We recommend that they be consistent across pages for future compatibility.

b. Busbar Voltage: Type the number for the busbar's voltage level, determined by the standard used in this installation. (Colours for busbar voltage levels are determined in the Project Color Change tool. See *Changing the Colours of Busbars and Circuit Breakers* on page 130 for more information.) To ensure that the Project Color Change tool operates correctly, voltages need to be consistent across the project.

   Do not connect two busbars with different voltage levels. The single line drawing will not display information correctly.

c. Line Active: From the drop-down menu, select the tag for the incoming voltage for device specified in the connected equipment. This can also be a Cicode function of expression (such as MyTag1 > 0).

   As an expression, use Line Active to link multiple-page drawings: on the incoming busbar on page 2, enter a line active statement that is equal to the tag used for the last device on page 1.

NOTE: If a field condition is available that can be used to indicate the actual busbar active state, it should be entered in the Line Active field.
Also, if there are transformers in the one-line, the busbar may not change colour smoothly after state changes. To minimize any delay in colouring busbars, you should, if possible, enter a valid expression in the Line Active field. For example, if you choose phase A voltage for line active of "MYCM4," and the incoming voltage is 440V, Line Active might read: MYCM4\MMXU1\PhV\phsA > 400. It is especially important to enter a line active expression for busbars that are directly upstream from the transformer.

5. Click OK to save the genie to the page.
6. If a message displays telling you that variable tags are not found, and asking if you want to add the unknown tags, this is not a problem. The genie is expecting to see a variable, but one may not exist in the equipment (for example, it could be looking for a "racked out" variable in a Sepam). Click No to this message.

   NOTE: To disable this message, go to the Project Editor page. Click Tools > Options. From the Options window, de-select "Prompt on tag not exist."

   The genie is added to the page.
7. Continue adding the genies to make up the graphics page.

   After you create the graphics page, you need to compile it. You can run the page to review its elements and correct any errors.

   NOTE: Do not drag genies off of the page. They will disappear.


© 2011 Schneider Electric All Rights Reserved

8. On the Graphics Page, click File > Compile.

   While the page is being compiled, the PowerLogic SCADA Compiler displays each file name as it is being compiled. A message then displays, telling you that compilation was successful.

9. If there are errors and/or warnings after the page is compiled, a message box displays, telling you the number of errors and warnings. You must understand all warnings and correct all of the errors. To begin, do the following:

   a. Click OK at the PowerLogic SCADA Compiler message.

      The Compile Errors window displays the first error.

   b. Note the error message. For more information on each error, click Help; note the Description for that Error Message.

   c. To correct the error, click the GoTo link.

      The appropriate window displays for you to correct that error.

10. Correct each error. Warnings do not need to be corrected in order to run the project; but they should be checked to see if they impact the project (such as a tag that is not defined).

11. If you delete any entries, click File > Pack to clean up the files.

12. Re-compile and verify that all of the errors have been addressed.

    NOTE: You can also view the Compile Errors window from the Project Editor (File > Compile Errors).

13. After all errors are corrected, run the project (File > Run).

    The Graphics page displays in the runtime environment. Note: Until you edit the Menu Configuration file, only basic tabs will display on the Graphics page.

14. Before you can view your one-line, you need to edit the Menu Configuration file, which controls the appearance of the graphics page in the runtime environment. See*Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121 for help.

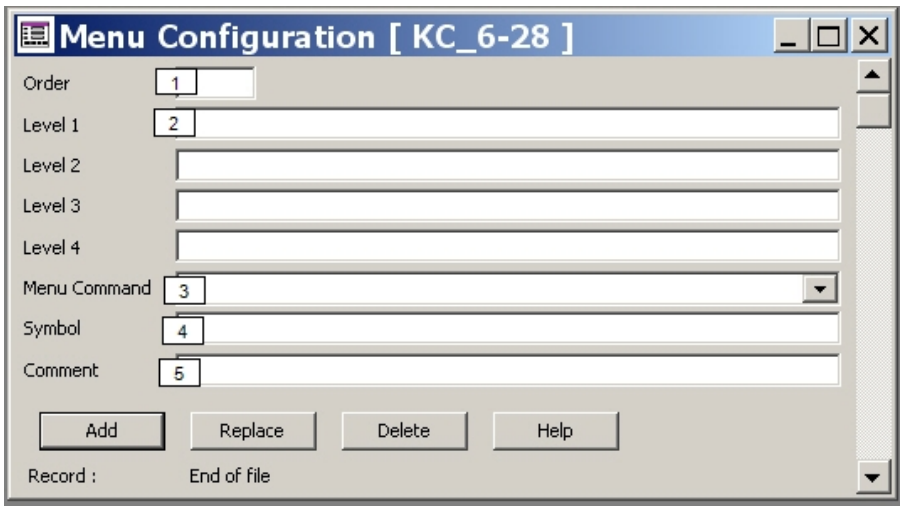
## Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)

The Menu Configuration form (System > Menu Configuration in Project Editor) edits Pagemenu.dbf in your project. This controls the runtime environment screen appearance: tabs and menus on the screen. By the entries you add there, you can also specify actions to be taken when an option is selected.

Because it takes much effort to create the items that display on a graphics page, you can use the menu settings from the Power SCADA PLS_Example project as a template for your new project template. From it, you can copy and paste settings into an individual project's menu configuration file.

The following illustrates a blank Menu Configuration file (see the table below for descriptions of the numbered fields):

© 2011 Schneider Electric All Rights Reserved




| Menu Item | Description |
|---|---|
| 1: Order | The relative position within the final graphics page. If you leave this field blank, the default value 0 is used. (64 characters max.) |
| 2: Levels 1 through 4 | These items establish the menu levels that will display. For example, you might use "Single Lines" for level 1, followed by the substation for level 2, and the graphic name for level 3. (Each line: 256 characters max.) |
| 3: Menu Command | The Cicode expression that you want to execute. Typically, you will use the "page display" command followed by the actual page you want to see. For example:<br>PLSPageDisplay("CB_IEC_1")<br>which displays the page CB_IEC_1. |
| 4: Symbol | Displays a defined image along with the description for that level.<br>Images must already be defined in the project/include project. They are specified in the format <library name>,<symbol name>. For example, in PLS_Example, the symbol used for the level 2 of Single Lines is Substation3, entered as **PLS_Icons.Substation3**.<br>Different menu levels are designed to be used with different symbol sizes for optimal display. For Level 1 items (tab), the recommended symbol size is 16 x 16 pixels. For Level 2 items, (buttons), the recommended symbol size is 32 x 32 pixels. Symbols are not displayed for menu items of Level 3 or beyond. |
| 5: Comment | You can use up to 128 characters to add a comment (will not display on screen). |

To copy information from the PLS_Example project, do the following:

1. Open the PLS_Example project. Then, from the Citect Project Editor, click System > Menu Configuration.

   The Menu Configuration for PLS_Example displays.
2. From Citect Explorer, change the project to your new project.
3. In the Citect Project Editor, click Menu Configuration.

   Now both Menu Configuration windows display.
4. To copy the entire record, you can use the Copy Record and Paste Record menu command. From the desired record in the PLS_Example project window, select the menu command Edit > Copy Record. Then go to the desired record in your project window, and select menu command Edit > Paste Record. Do further editing as required. Alternatively, you can selectively copy information from the source record to your project, line by line.
5. To save the menu configuration record in your project, choose either the "Replace" or "Add" button. Compile the project; then run it again.

   The Graphics page in the runtime environment of your project will now display according to the settings you have made.


© 2011 Schneider Electric All Rights Reserved

## Creating and Editing Genies

The easiest way to create a new genie is to make a copy of a similar genie from the standard PowerLogic SCADA library and edit it in your project. We recommend, when adding layers, that you keep the dimensions of the new layers the same as the original. For a list of all of the standard PowerLogic genies, including all of the smaller parts that you could use to create genies, see *Appendix 1: Default Genie Library* on page 163.

When you modify a genie, the modifications will be in effect for all instances of that genie in the project.

### To create a custom genie, follow these steps:

1. Open the genie that you want to use as a template: From Citect Graphics Builder, click File > Open. Click the Genie tab, then select the library and the template genie. Click OK.
2. Save the genie with a new name in your local project.
3. To separate the genie into layers, click the genie and drag a layer to the bottom right.
4. Repeat the process to pull all of the layers apart.

   Each layer is a sub-genie that controls a different aspect of the overall genie.
5. Make the changes that you wish to the genie.

   NOTE: Be careful to maintain the same dimensions for any new layers that you create.
6. To re-assemble the genie, draw a marquis around all of the parts; then click Arrange > Align, and choose Top and Left. Click OK.

The newly created genie, when applied to a page, will display with the generic input form. To create a customized form similar to those found with the default genies, you must create a new .FRM file. Examples are found in the PLS_Include project directory.

You can rotate any of the genies.

### To create a custom symbol for a custom genie, follow these steps:

PowerLogic SCADA genies are complex PowerLogic SCADA genies with the added ability to animate colour during runtime (showing changing states). These genies are comprised of multiple symbols that have the same attributes (all will work with Project Color Change tool, all call the same Cicode function, etc.).

1. Create the new symbol.
2. Configure the symbol to work with the Project Color Change tool: from the Symbol properties screen, choose the Access tab. In the Description field, enter <PwrLineColor>.
3. Configure the symbol to display the customized colours in the runtime environment by calling the *PLSAnimBusbar()* function, which is provided in PowerLogic SCADA. This function must be called from the Multi-State Condition area of the Symbol properties window.

   Each multi-state symbol can display up to seven colors (see *Changing the Colours of Busbars and Circuit Breakers* on page 130 for details). These symbols are labeled in binary with ABC and are passed to the *PLSAnimBusbar()* function as the first parameter. The second parameter is the busbar number that determines the animated colour. Always use the %variable% notation. Using this notation permits you to reuse genies.

   See the default symbols in the PowerLogic SCADA libraries that begin with PLS_.

© 2011 Schneider Electric All Rights Reserved

4. Save the symbol and use it in a custom genie.

## Deleting Information from PowerLogic SCADA

If you find you need to delete any data that you have entered (clusters, servers, genies, etc.), follow the normal procedure to delete the data, then use the Pack command to completely delete it. To do this, click File > Pack on the Project Editor.


© 2011 Schneider Electric All Rights Reserved

# Working with Alarms

In this section, you will find these topics:



## Alarms Overview

This section discusses two alarm types: time stamped analog and time stamped digital. To access the alarms, from Citect Explorer, select the project folder, then click Alarms. In the right-hand pane, the alarm types display. Double-click the one you want to view/edit.

### PC-Based Alarms

PC-based alarm tags are added in the Profile Editor, when adding each device profile. See *Adding, Editing, or Deleting a Device Profile* on page 52 for instructions. For instructions on entering setpoints and delays, see *Adding Setpoints and Delays* on page 125.

### Onboard Alarms

If onboard alarms have been configured in a supported device, you can use the Profile Editor to map these alarms to digital time-stamped alarms in PowerLogic SCADA.

You cannot configure new onboard alarms from PowerLogic SCADA. You must add the alarm at the device, then you can create the alarm tag for it here. See *Adding an Onboard Alarm Tag* on page 128.

## Adding Setpoints and Delays

NOTE: Any time you change setpoints, you should immediately restart the project. Otherwise, setpoints will not be properly read (they will be truncated and either rounded down or up to a whole integer).

There are two ways to add setpoints and delays for analog alarms. Before you enter setpoints and delays, ensure that you have configured the alarm server so that Publish Alarm Properties is set to TRUE.

- From the Analog Alarms window (accessible from the Project Explorer or Project Editor screens), you can type the setpoint and delay values for each alarm.
- When in the runtime environment, you can edit setpoints/delays that were set by the method above. See *Viewing Waveforms* on page 151 for instructions.

Also, set the following parameter to allow persisting of alarm parameters at runtime.

[Alarm] UseConfigLimits = 1

© 2011 Schneider Electric All Rights Reserved

## Setting Up an Alarm Based on an Enumeration

For instructions on defining an enumeration in the Profile Editor, see *Defining an Enumeration* on page 79.

An example of an enumeration alarm is:

0 = unknown

1 = good

2 = warning

3 = alarm

To add an alarm that is based on an enumeration, follow these steps:

1. Open the analog alarm in PowerLogic SCADA.
2. To alarm on states 0, 2, and 3:

   Set Low = 1 (if the value < 1, the alarm indicates an unknown state)

   Set High = 1 (if the value > 1, the alarm indicates a warning)

   Set High High = 2 (if the value > 2, the alarm indicates an alarm)
3. In the Category field, ensure that the correct alarm level is entered (_PLSALM_HIGH, _PLSALM_MEDIUM, _PLSALM_LOW, _PLSALM_EVENT).
4. Replace the alarm.

## Changing an Alarm Severity

To change the severity of an alarm, follow these steps:

1. Open the analog alarm in PowerLogic SCADA.
2. In the Category field, ensure that the correct alarm level is entered (_PLSALM_HIGH, _PLSALM_MEDIUM, _PLSALM_LOW, _PLSALM_EVENT).
3. Replace the alarm.

## Enabling Waveforms for Onboard Alarms

To enable waveforms, making them available for viewing in the runtime environment, follow these steps:

1. At the device, or via the meter configuration software (PMCU), add the alarm and enable the automatic capture of a waveform when the alarm occurs.
2. In the Profile Editor, on the Create Device Profiles tab, for the same alarm you added in PMCU, check the Waveform box.

   When this alarm occurs at the device, the waveform will be captured. The files will be transmitted to PowerLogic SCADA and will be available for viewing. The amount of time this takes depends on the number of I/O servers you have and the number of serial devices on a chain. On a very large system with numerous serial devices, this could take as much as an hour.

   You can view the waveform from the Alarm Log in the runtime environment. See *Using the Equipment Pop-Up Page* on page 150 for instructions.

## Setting Parameters for Event Log Length and Historical Logging of Events

You can use two parameters to determine the maximum number of entries in the Event Log and whether you want to log entries after they are FIFO'd out of the Event Log.


© 2011 Schneider Electric All Rights Reserved

### Event Storage: [Alarm]SummaryLength Parameter

The maximum number of alarms that be stored is controlled by the Alarm Summary length parameter, which defines the maximum number of alarm summary entries (Event Log entries) that can be held in memory. You can view these alarm summary entries on the Alarm Log page. Each event requires 256 bytes of memory, plus the length of the comment. 32,000 entries will require at least 8 MB of memory. If you have many events, you should ensure that there is enough memory to store them in RAM.

The default value is 5000.

NOTE: When the value is set to a number greater than 1000 for a multiple-cluster system, the alarm log might not display correctly. The list of alarm history that displays on a client might be shorter than the actual history stored on the alarm server. To avoid this problem, do one or more of the following:

- Set alarm filtering in the alarm viewer to reduce the number of alarms that are returned by the server.
- Only support a one-cluster system.
- If a multiple-cluster system is necessary, display a separate alarm page for each cluster.

### Event Logging: [PLSEventLog] Parameter

Use this parameter to determine whether you will log Event Log entries and the file format that will used for logging. If you choose to log, you must specify the file format: XML or CSV.

**[PLSEventLog]**

FlleFormat = 0 = no file logging, 1 = CSV format, 2 = XML format (default = 0)

EntryCount number of entries logged per disk file (1-3200; default = 250)

ScanRate time in seconds between writing to file, when waiting for a new event (1 to 3200; default = 10); may be ignored if the entry count is reached or if a large number of alarms or events is triggered

The filename will have this format:

<Project Name>_<Cluster Name>_PLSEventLog.YYMMDDHHMMSS.xxx (where xxx is the file format selected). The log files will be stored in the Data folder.

[Path]

PLSEventLog=c:\SharedData\Alarm

To specify the alarm fields to be logged, use:

**[AlarmFormat]**

EventLog=OnDate | Date, OnTimeMS | Time, Custom1 | Equipment, Name | Description,
SumState | State | Custom2 | Location, UserName | User

**Format]**

Summary =

Alarm =

Equipment =

LastAlarm =

**[AlarmHeading]**

<FieldName> = <Display Name>

© 2011 Schneider Electric All Rights Reserved

## Adding an Onboard Alarm Tag

When a device onboard alarm has not been included in PowerLogic SCADA, you can add it from the Profile Editor. You need to follow these steps to include the device's unique identifier. Otherwise, the alarm will not annunciate in the Graphics page.

NOTE: You can only add onboard alarms for devices using the CM4, PM8, Micrologic, or Sepam drivers. CM4, PM8, and Micrologic unique IDs must be decimal; SEPAM unique IDs must be hexadecimal.

1. From the device, obtain the unique identifier for this alarm. Additionally, for MicroLogic, you need to include the unique sub-identifier.
2. You also need the file number in which alarms on stored on the device.
3. From the Profile Editor, add the onboard alarm (see *Onboard Alarm Tag Addresses* on page 47 in Editing Tag Addresses.

## Setting Up Audible Alarms

You can use a variety of Windows wave files for audible alarms. To enable this feature, follow these steps:

1. Define the alarm sound to be used and the repeat interval for each priority in the alarm you want to be audible. Enter the following information, either in the project parameters (Citect Project Editor > System > Parameters) or in the Citect.ini file:

   a. [Alarm]

   b. Sound<priority>=<wave file name>

   c. Sound<priority>Interval=<repeating interval in milliseconds>

   If the device is added through the Profile Wizard, the alarm priority will be 1, 2, and 3 for _PLSALM_HIGH, _PLSALM_MEDIUM, _PLSALM_LOW alarms respectively.

   The user can define specific wave files for the sounds, PowerLogic SCADA also supports the following Windows operating system sounds:

   SystemAsterisk

   SystemExclamation

   SystemQuestion

   SystemDefault

   SystemHand

   SystemExit

   SystemStart
2. After alarms are set up: When an alarm occurs, its specified alarm sound will play continually according to the specified interval. The alarm sound will stop when either:

   — the user clicks Silence Alarm on the alarm page

   — the alarm is acknowledged


© 2011 Schneider Electric All Rights Reserved

# Restoring and Backing Up Projects

In this section, you will find these topics:



## Restoring a Project

To restore a project, overwriting its current settings, follow these steps:

1. From the Citect Explorer, select the project you want to restore; click Tools > Restore.
2. In the Restore from box, browse to the location of the file you will use to restore.
3. In the To box, check Current Project.
4. In the Options box, check *Configuration files* to restore backed up INI files and the TimeSyncConfig.xml file (used to store time synchronization settings). If you backed up the sub-directories under the project, the directories will be listed under *Select sub-directories to restore*. You can restore all or no sub-directories, or you can select specific sub-directories to restore.
5. Click OK.

## PowerLogic SCADA Backup

Use this feature to back up a Citect project file. To back up a Profile Editor project file, see *Profile Editor Export* on page 71.

1. From the Citect Explorer, click Tools > Backup.
2. At the Backup Project window, select the project you want to back up
3. Browse to the location where you want to store the backup file.
4. In the Options box, check “Save configuration files.” This saves the citect.ini file.
5. Click OK.

The backup .CTZ file is written to the following location:This is a Citect Zip file; you can open it with WinZip.

© 2011 Schneider Electric All Rights Reserved

# The Project Color Change Tool

Use this tool to assign colours for busbars and circuit breakers that are generated by genies in graphics pages. You will set separate colours for each project in your system. There are seven states (Error, Intermediate, etc.), for which you can assign colours. You can also select a second “flash” colour for any state; when the busbar or circuit breaker is in that state, the colour will alternate between the two colours.

It is best to set the project colours before you add genies to the project. A large number of standard genies (beginning with PLS_) are already available; however, you can create a custom genie and add it to the project. If you choose to add genies, follow the instructions in *Adding a Custom Genie for Use in the Color Change Tool* on page 130.

To begin changing colours, see *Changing the Colours of Busbars and Circuit Breakers* on page 130.

## Adding a Custom Genie for Use in the Color Change Tool

To be able to change colours for a project, all of its genies must be set up in the manner described below. The standard PowerLogic libraries of genies already have this custom setting, but if you need to create a genie or use one that is not in the PowerLogic libraries, follow these steps:

1. From Citect Explorer, choose the project for which you want to set/change colors.
2. From the Graphics Builder, click File > New > Genie.
3. On the Objects toolbar, click the object you want to draw or insert, then click on the graphics page.
4. On the Properties page, set the type to *Multi-state* (this will be on one of two tabs: Appearance or Fill). Set its conditions and states, then click Apply. If you need help adding conditions and states, click Help.

   NOTE: The State colors that are shown on this tab are what you will be changing in the Project Color Change Tool.
5. Click the Access tab. In the Description field, type:

   <PwrLineColor>
6. Click OK to save the genie.
7. Open the graphics page, and drag and drop the genie onto the page.
8. To run the Project Color Change Tool, you need to close any open graphics pages.

## Changing the Colours of Busbars and Circuit Breakers

Open the Project Color Change tool: from the Citect Editor screen, click Tools > Project Color Change Tool. Then follow these steps:

1. If a prompt displays telling you that Citect.ini location has changed, click OK. There is no need to locate the file.

   Colours will be changed for the genies in a project only if these conditions are true:

   — The genie was set up with a multi-state type and that it has <PwrLineColor> as its description on the Access tab.
   — The selected project has supported genies.
   — The selected project's direct include projects have supported genies.
   — The selected project's direct include project has an include project with supported genies, and the direct include project is not hidden.

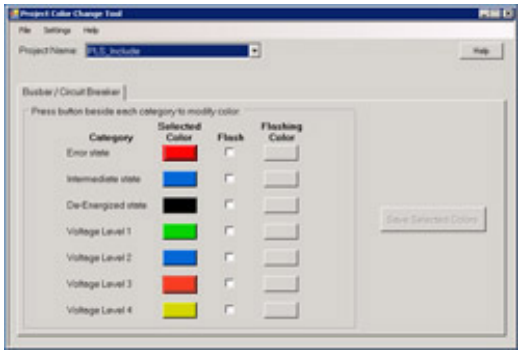
   NOTE: If no projects display, one of two things is true: either your projects are “hidden” (in the Include folder) or the genies in the project do not have the <PwrLineColor> setting described in *Adding a Custom Genie for Use in the Color*


© 2011 Schneider Electric All Rights Reserved

*Change Tool* on page 130. If any genie on the page does not change colour, check to be sure that you have added the setting for that genie.

2. At the message telling you that PowerLogic SCADA is running and will be hidden, click OK.

   The Project Color Change Tool displays (the colours you see may differ):



3. From the Project Name drop-down menu, choose the project for which you want to assign colours.
4. Click the selected colour for the first state or condition that you want to change:

| Category | State of the Busbar or Circuit Breaker | PowerLogic SCADA Multi-State Colours* |
|---|---|---|
| Error State | The circuit breaker has lost communication. | ABC / **A**BC (colours 1 and 2) |
| Intermediate state | The circuit breaker is between states, e.g., being racked in or racked out. | A**B**C (colour 3) |
| De-Energized state | The circuit breaker is de-energized or inactive. | **AB**C (colour 4) |
| Voltage level 0 | These levels correspond to the four voltage levels that (busbar voltage level), entered when adding a busbar to a drawing. These correspond to the levels being used by the standard that applies for each installation. | AB**C** (colour 5) |
| Voltage level 1 | | **ABC** (colour 6) |
| Voltage level 2 | | A**BC** (colour 7) |
| Voltage level 3 | | **ABC** (colour 8) |

* The Symbol Properties screen is used when creating conditions for multi-state colours. See *Creating and Editing Genies* on page 123 for details on animating a custom genie.

NOTE: Although the Color Change Tool uses only numbers 1 through 8, Citect allows you to use 0. However, using 0 produces unstable colours in graphics. Thus, we recommend that you NOT use colour 0.

5. From the Windows Color screen, choose the new colour, then click OK.
6. If you want busbars and circuit breakers in the Graphics page to flash when they are in a particular state, click Flash for that state, then add the “flash” colour. For example, if you set the error state colour to red, you might add a flash colour of green.

   When the graphics page is opened for viewing and busbars or circuit breakers are in that state, they will flash continuously between the two colours.
7. Continue to change state colours, or click Save Selected Colors to save your colour changes.
8. To update graphics pages with the new colours, go to the Graphics Builder. Click Tools > Update Pages.

The colours will be used for busbars and circuit breakers in all genies for this project (as long as the genie has *Multi-state* as its type and has the description of <PwrLineColor>.

© 2011 Schneider Electric All Rights Reserved

# Using Cicode to Customize a Project

Cicode is a programming language designed for use in PowerLogic SCADA to monitor and control plant equipment. It is a structured language similar to Visual Basic or 'C'. You need no previous programming experience to use it. However, it is assumed that you will have received Cicode training from PowerLogic SCADA before you attempt to use Cicode.

Using Cicode, you can access all real-time data (variables) in the PowerLogic SCADA project, and all PowerLogic SCADA facilities: variable tags, alarms, trends, reports, and so on. You can use Cicode to interface to various facilities on the computer, such as the operating system and communication ports. Cicode supports advanced features including pre-empted multitasking, multi threads, and remote procedure calls.

For information about driver-specific INI parameters that you can configure, see *Appendix 2: Using Citect INI Parameters* on page 171 .

For information about other parameters, see the CicodeReference help file, located in Program Files\Schneider Electric\PowerLogic SCADA 7.20\Bin.


© 2011 Schneider Electric All Rights Reserved

# Customizing PowerLogic SCADA Default Behaviours

In this section, you will find these topics:



To learn about the PowerLogic SCADA parameters that you can use to configure drivers, see *Appendix 2: Using Citect INI Parameters* on page 171.

## Time Zone Settings

The system allows you to interact with devices located in a different time zone than the PowerSCADA I/O Server. To do this, the system converts any alarm/waveform timestamp as well as the actual time sent within the Set Time command from / to the local time zone. In order to take into account daylight savings properly, the Windows time zones database is used. Thus, time zone names must be taken directly from this database (case-insensitive), otherwise the system will default to using the I/O server's local time zone. The Windows time zone database is located in the Windows registry in HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\Time Zones\. Examples of time zone names are

- AUS Central Standard Time
- China Standard Time
- Pacific Standard Time (Mexico)

Time zones for remote devices can be specified on two levels:

1. Use section [ProtocolName.ClusterName.PortName.IODeviceName] to specify the time zone for a particular device, e.g.

   [PLOGIC870.Cluster1.PM870_Port.PM870_Device1]

   Timezone = Singapore Standard Time
2. Use general section [POWERLOGICCORE] to specify the time zone for all devices, e.g.

   [POWERLOGICCORE]

   Timezone = Mountain Standard Time

The device-specific time zone specification takes precedence. In other words, if both of the above examples are present in the Citect.ini file, the PM870_Device1 would be located in "Singapore Standard Time" time zone, and all the other I/O devices in the project would be located in "Mountain Standard Time" time zone.

If there is no time zone specification (or if it does not match the time zone from Windows database), the device would be in the same time zone as the machine where the I/O Server is running; thus, no time conversion will be done.

If only the first of the above examples is present within the Citect.ini file, the PM870_Device1 would be located in "Singapore Standard Time," and all the other devices use the current local time zone.

## Time Synchronization

Current time can be sent to the corresponding device by means of Set Time command or (in case of Sepam) by writing directly to the corresponding registers within the

© 2011 Schneider Electric All Rights Reserved

device. In addition to the manual procedure, this process can be scheduled to occur periodically (using PowerLogic SCADA events).

Non-manual time synchronization causes the Set Time command to be sent automatically, based on a device state or event originating from within the device.

Automatic time synchronization applies only to Micrologic and PM devices and takes place based on the following rules:

1. For Micrologic devices, the value of the top-most bit of the register 679 is examined (for both the Circuit Breaker Manager and the Chassis Manager). If the bit is equal to 1, it means that the device is out of sync and needs to be synchronized.
2. For PM devices, an alarm 50700 (“Unary Power Up / Reset”) indicates that the device needs to be synchronized. In addition, bit 6 of register 3055 of the device is examined. If this bit is equal to 1, the device has a real-time clock; so automatic time synchronization should never take place.

## Trend Tag Scan Intervals

When you select a trend tag for a device profile (Add/Edit Device Profile screen), the tag will be scanned at the “long” interval” (every 15 minutes, with FIFO storage of 12 months); but certain trend tags have an additional “short” scan interval. This interval is set by default at 5 seconds, with FIFO storage of two weeks.

The default tags are: Current A, Current B, Current C, Voltage A-B, Voltage B-C, Voltage C-A, Power Factor Total, Apparent Power Total, Reactive Power Total, Real Power Total, and Frequency. When you choose one of these tags for trending, you will get both long and short interval trending. The long interval trend will use the trend tag name from the Profile Editor. The short interval trend tag will have the same name as the long tag with an “s” appended to it.

You can edit the *Profile Editor.exe.config* file to add or delete tags that will have short scan intervals, and to change the short scan interval for all of the tags that are listed.

To edit short scan interval settings, follow these steps:

1. Locate and open *Profile Editor.exe.config* in Notepad:

   [Project Drive]\Documents and Settings\ All Users\Application Data\Schneider Electric\Profile Editor\v7.20

   Vista operating systems: [Project Drive]\ProgramData\Schneider Electric\PowerLogic\Profile Editor\v7.20)
2. To change the short scan interval, scroll the "TrendShortIntervalSamplePeriod" setting. The default value is 00:00:05, or five seconds (HH:MM:SS). Changing this rate will change the interval for all of the tags that are listed in the setting in step 3.
3. To change the tags that are included in the short scan interval, scroll to the "TrendShortIntervalTags" setting. The numbers listed (defaults: 1003,1004,1005,1050,1046,1042,1014,1015,1016,1001,1034) are the tag IDs. You can add or delete tags. Tag IDs are listed on the Define Device Type Tags tab (when the Advanced Properties option checked).

   NOTE: If you choose a device that includes the tags in this list, you will always have these short scan interval tags included.

For example, if you wanted to change the scan interval to ten seconds and add Overcurrent A for a CM4000, you would edit these two lines in this way:

"TrendShortIntervalSamplePeriod" value="00:00:10"

"TrendShortIntervalTags"
value="1003,1004,1005,1050,1046,1042,1014,1015,1016,1001,1034,19"


© 2011 Schneider Electric All Rights Reserved

## Calculating Disk Storage for Trends

There are two methods of calculating disk space usage for trends: scaled and floating point. The Profile Editor uses floating point by default. For more information on these calculations, see Calculating Disk Storage in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).

© 2011 Schneider Electric All Rights Reserved

# Waveform Management

This chapter discusses how waveforms are stored and associated with alarms. In this section, you will find these topics:




© 2011 Schneider Electric All Rights Reserved

# Waveform Storage

Waveform records are organized within devices into files. These files are periodically checked for and downloaded as they appear on the device. When downloaded, on the PowerLogic SCADA I/O server the files are converted into a Comtrade format and then stored in a hierarchical fashion.

A single waveform will be stored as follows:

<Waveform DB root>\<ClusterName>\<IODeviceName>\Waveforms\<UTCTimestamp>.CFG

<Waveform DBroot>\<ClusterName>\<IODeviceName>\Waveforms\ <UTCTimestamp>.DAT

For example,

C:\Data\Cluster1\Sepam_IODev\Waveforms\

DST_00000000001203566197_0000000511_utc.CFG

DST_00000000001203566197_0000000511_utc.DAT

Waveform DB root path is configured in the WaveformDB configuration section.

NOTE: in case of redundant I/O devices, only the name of the primary I/O device will be used when waveform storage path is constructed.

The CFG file is a **Comtrade configuration file**, and the DAT file is the **Comtrade data file**. Within the CFG file is a timestamp that reflects the device time start time of the waveform (this time is not adjusted to the I/O Server time zone or daylight saving, but it is stored per the device configuration). The file name has the UTC time in seconds since 1970 of the waveform.

The prefix of waveform file name reflects the type of the waveform. Currently, waveforms of the following types are supported:

| | |
|---|---|
| DST_ | Disturbance waveform |
| ADT_ | Adaptive waveform |
| SST_ | Steady state waveform |

If it is detected that the waveform data file has changed while it is being downloaded, the file gets discarded and is not stored on the IOServer.

© 2011 Schneider Electric All Rights Reserved

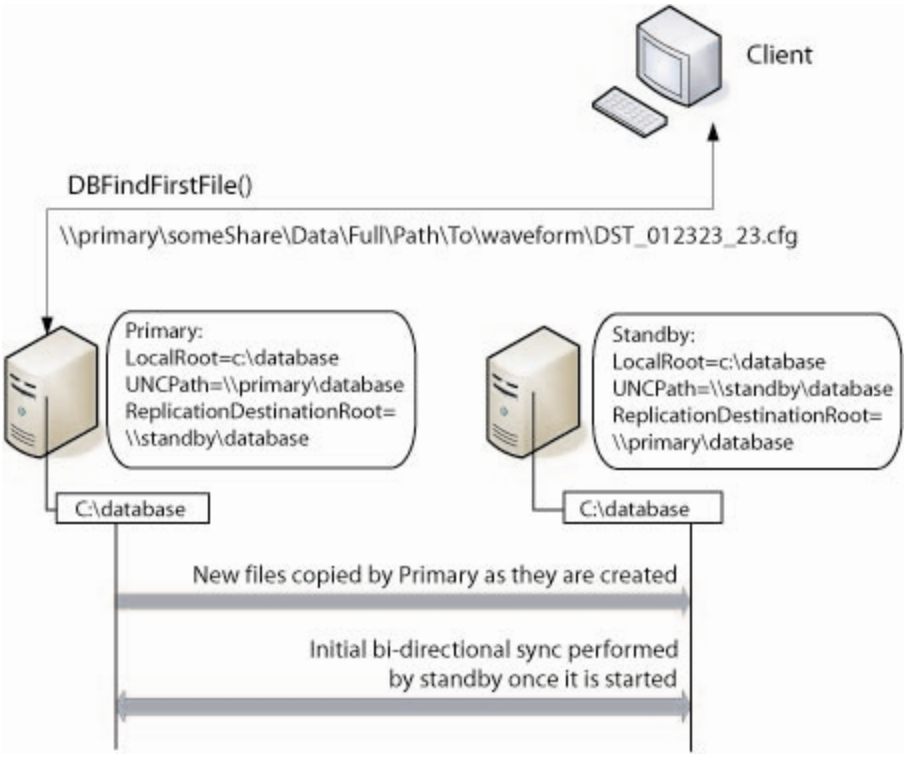
# Waveform Database and Special Waveform Tags

PowerLogic SCADA allows you to browse the waveform database for specific I/O devices. Search for all waveforms within certain time frame is also supported, allowing you to search for all waveforms that could be linked with a given alarm. When you perform this search, a list of all matching waveforms displays. If there are multiple waveforms in the list, you can select the waveform you want to view.

In addition, there are two special digital waveform tags defined (0 = FALSE, 1 = TRUE):

- **WaveformDownloading**: indicates whether a waveform file is currently being downloaded
- **WaveformCollectionEnabled**: indicates whether the waveform collection is enabled at all

The figure below illustrates a configuration example and replication and linkage processes.





© 2011 Schneider Electric All Rights Reserved

# Graphics: Using the Runtime Environment

This chapter discusses the Graphics page in the runtime environment. In this section, you will find these topics:



© 2011 Schneider Electric All Rights Reserved

# Launching and Closing the Runtime Environment

**Launching Runtime**: To launch the runtime environment, click Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Runtime. Or, from the Project Explorer, Project Editor, or Graphics Builder design-time page, click File > Run.

**Closing Runtime**: There are two ways to shut down the runtime environment:

- You can click the “X” in the upper right corner of the screen. You must enter a user name and password. (It is important to define at least one user in your project.)
- From the computer taskbar, click the Runtime icon:

  This displays the Runtime Manager screen. Click Shutdown All. There is no required user name/password. Unless you disable this feature on the taskbar, users can shut down the runtime environment without having to enter any information.


© 2011 Schneider Electric All Rights Reserved

# Viewing the Graphics Page in the Runtime Environment

In the runtime environment, the Graphics Page displays with the appearance that is determined by the Menu Configuration file for this project. If you need to change the appearance of tabs and menus, see *Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121.

In this section, you will find these topics:



## Logging On to the Runtime Environment

In the upper left-hand corner, click the Login link. Then enter your User ID and password. The features that are available will vary, depending on the user level.

## Viewing the Interface

The runtime environment page includes a variety of tabs. If your page is created based on the Normal template from the pls_include_1 library, you will see the following:

In the upper left corner of the screen, the project name is listed. Your logon name is below that. At the top of the page, there are left and right navigation arrows that allow you to go to previously visited pages.

These two arrows allow you to go back and forward one page in your navigation history. To see the history of visited pages, click the small down arrow next to the right arrow. This displays a listing of visited pages (the current page is checked). To jump to a page in this list, click it in the menu.


Just beneath the left and right arrows is the alarm banner. It lists the last five active alarms.

Beneath the alarm banner is a tabbed-style menu. Its contents are determined by the information entered in the Menu Configuration tool: *Using Menu Configuration to Edit Pagemenu.dbf (Changing the Graphics Page Appearance)* on page 121. These tabs display in two rows:

- The upper row is typically used for organizing pages into several topics (or tabs). A typical system would include topics for single lines, alarms/events, analysis (for trends), and system supervision (allows you to view the network connection topics).
- The lower row lists the links/pages under the topic that is currently selected in the upper row. If you select the single lines topic on the upper row, the lower row displays all of the links to individual single line pages.

If there are more links available than the ones that fit on the page, a small arrow displays at the right hand side of the row. Click the arrow to display a pop-up menu of the remaining links. Click a link in the menu to shift the contents of the row to make it visible for selection.

NOTE: When viewing the single lines: if the busbars and circuit breakers do not display with the expected colors, it could be that a custom genie is not set up correctly. If you

© 2011 Schneider Electric All Rights Reserved

are using a custom genie, verify that <PwrLineColor> has not been left out. See *Adding a Custom Genie for Use in the Color Change Tool* on page 130 for details on customizing genies and displaying colors.

### Communications Loss

When there is a communication loss for a device, the genie or any part of the genie on the one-line page should have cross-hatches (gray dots) over the affected area, and a communication loss (CL) indication displays on the genie. An alarm should also annunciate. The color state before communication was lost will remain unchanged.

**However, the indication of loss of communications does not filter through the entire bus animation: the downstream part of the drawing may still appear as if communication is working. When any part of a one-line drawing loses communication, do not continue to trust downstream readings until you address the loss of communication.**

## Viewing the Alarms/Events Page

See also: *Using the Equipment Pop-Up Page* on page 150

To view the alarms or events, click the Alarms/Events tab; then select the tab for the Alarm Log or Event Log. The Event Log displays all alarms and events that have occurred. The Alarms Log displays enabled alarms.

### The Event Log

The Event Log displays all alarms and events that have occurred.

### The Alarm Log

The Alarm Log displays active enabled alarms. To sort on any of the columns, click the column title.

To filter the alarms that display, click Filter, just above the Date column (see instructions on using the filter option in *Filtering and Freezing Log Views* on page 147). From the Advanced Alarm Filter window, you can filter by date range, by text matches for various attributes, or by alarm type.

To freeze the view of the current log (this is also true for unacknowledged or disabled alarms), click Freeze in upper right corner of the screen. (Note that the button name changes to Unfreeze.) No additional alarms will display and none will drop out until you click Unfreeze.

Each alarm provides additional options. To view these options, right-click the alarm. Then you can:

- acknowledge or disable the alarm
- view alarm detail (similar to the genie status page in the single lines of the runtime environment)
- view waveforms: If this option does not display, there are no waveforms for this alarm.) Waveforms can display only if the device is set to “acquire on event,” and the waveform option is checked in the Profile Editor (see *Enabling Waveforms* on page 51).

  When the waveform is available for viewing, it displays when you click this link. Waveforms must be correctly set up before they will display. See *Enabling Waveforms for Onboard Alarms* on page 126 for more information. See *Viewing Waveforms* on page 151 for instructions on viewing waveforms.

  When you select the waveform option, you may see a message telling you “please try again after waveform has been acquired.” This means one of two things:


© 2011 Schneider Electric All Rights Reserved

— the alarm has been acquired at the device, but it has not yet been passed to PowerLogic SCADA

— the device was not set to acquire a waveform, and the waveform option was checked in the Profile Editor

NOTE: If there are multiple waveforms captures for this alarm, and if there is a disturbance waveform, it is the only one that is available here. If there are both an adaptive and transient, but no disturbance, the one with the earliest time stamp displays.

### Unacknowledged Alarms and Disabled Alarms

As with the Alarm Log, these logs display either unacknowledged alarms or disabled alarms. The sort and filter options operate as they do in the Alarm Log.

### Alarm and Events Logging

Alarms from the Event Log can be saved to a file on the alarm server, thus protecting them from being lost when the FIFO size is passed. This feature is disabled by default, but it can be enabled by setting the FileFormat INI parameter.

[PLSEventLog] FileFormat: Determines the file format to be used for logging alarm/event data to disk files.

Allowable Values:

0 - (Disable)

1 - (CSV)

2 – (XML)

Default value: 0

### Sorting Logs

To sort a log, displaying information according to one of its columns, double-click the title. To sort the date/time, double-click the Date column; the times within that date will automatically display, newest to oldest. When you exit the log, the settings are lost. When you re-enter the log, you will need to perform the sort again.

The alarms display in this order: active alarms in chronological order followed by inactive alarms in chronological order.

### Filtering and Freezing Information

To filter the information that displays, click Filter, just above the Date column (see *Filtering and Freezing Log Views* on page 147 for more information). From the Advanced Alarm Filter window, you can filter by date range, by text matches for various attributes, or by alarm type.

To freeze the view of the current log, click Freeze, in the upper right corner of the screen. No additional alarms will display, and none will drop out, until you click Unfreeze.

### Alarm Viewing Options

Each alarm provides additional options. To view these options, right-click the alarm. Then you can:

- acknowledge or disable the alarm
- view alarm detail (similar to the genie status page in the single lines of the runtime environment)
- view waveforms: If this option does not display, there are no waveforms for this alarm.) Waveforms can display only if the device is set to “acquire on event,” and

© 2011 Schneider Electric All Rights Reserved 

the waveform option is checked in the Profile Editor (see *Enabling Waveforms* on page 51).

Choose this option to display waveforms that were captured by this device in the 24 hours before the time of the alarm. You can change the time range from the Search Waveform popup that displays. Choose a date range, or click All Available to display everything for that device in the waveforms directory. Click OK to display the list of waveforms that match the time range. Choose the waveform you want to view, and click View.

For information about the options available on the waveform, see Viewing Waveforms in *Using the Equipment Pop-Up Page* on page 150.

Waveforms must be correctly set up before they will display. See *Enabling Waveforms for Onboard Alarms* on page 126 for more information. See *Using the Equipment Pop-Up Page* on page 150 for instructions on viewing waveforms from the genie status page.

When you select the waveform option, you may see a message telling you “please try again after waveform has been acquired.” This means one of two things:

— the alarm has been acquired at the device, but it has not yet been passed to PowerLogic SCADA

— the device was not set to acquire a waveform, and the waveform option was checked in the Profile Editor

If there are multiple waveforms captures for this alarm, and if there is a disturbance waveform, it is the only one that is available here. If there are both an adaptive and transient, but no disturbance, the one with the earliest time stamp displays.

See *Changing the Display Name in an Alarm or Event Log* on page 146 for more information.

### Viewing, Printing, and Export Options

Each of the logs includes these navigation buttons:

| Button | Description |
|---|---|
|  | Acknowledge Current Page of Alarms: Click to acknowledge all of the alarms that display on the current page.<br>**Note**: You can acknowledge individual alarms in this way: On the left side of the alarm that you want to acknowledge, right-click, then choose Acknowledge. On a touch screen, tap the left side of the alarm to display the menu, then tap "Acknowledge." |
|  | Silence Alarms: Click to silence all active alarms. This does not clear unacknowledged alarms or make alarms inactive; it only stops the audible portion of the alarm. |
|  | Print/Export Alarms: Click to begin printing or exporting part or all of the log. Select All or the number of pages, then choose whether to print or export (to HTML file, which can then be opened in Excel or OpenOffice). When printing, the default location is:<br>Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\Data<br>Vista operating systems: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\Data<br>**Notes**: When printing: To avoid truncation of data, choose the Landscape orientation.<br>When using Internet Explorer 8 and a dot matrix printer, you might have problems with overlapping columns in the printout. To solve this, either switch to Internet Explorer 7 or select a "square" matrix (e.g., 180 x 180 DPI). |
|  | Save Columns: Click to save the configuration of columns that you have selected after you close the Alarm Log. You must be an administrator to do this. |
|  | Add Columns: Click to add additional columns. Choose from the list. Columns will be placed on the right side of the table. See *Event/Alarm Log Columns Table* on page 145 for a list of columns an names. |


© 2011 Schneider Electric All Rights Reserved

| Button | Description |
|---|---|
|  | Auto Fit Columns: Click to correct the alignment of columns, to ensure that they all fit on the page. |
|  | Page Number: Display only; this is the page number that you are viewing. |
|  | Page To Top and Page Up: Click to either go to the top of the log or to move up one page. |
|  | Page Down: Click to move down one page.<br>**Note**: If the page is full, but there are no more alarms, clicking here will display a blank page. Click it again to return to the previous (full) page. |

## When Alarms do not Display Correctly

Alarms may display incorrectly for a variety of reasons. The following table lists some common issues and resolutions.

| Issue | Cause | Resolution |
|---|---|---|
| Alarm Log and Event Log do not display any data. | If there are two alarm servers, primary and redundant (standby), they may be synchronizing. This causes data to display slowly. | Data will display; but it could take several minutes. |
| Alarms display in Alarm Log, but not in Event Log or Banner | The missing alarm(s) were triggered while the runtime graphics page was not running. | These alarms will only display in the Alarm Log unless they are triggered again while the runtime graphics page is running. This will only affect alarms that were triggered before the runtime screen was running. |
| PC-based and onboard alarms do not appear or disappear as expected. | This is due to the difference between way the two alarm types are handled:<br>When an alarm is enabled, the system processes alarms for that tag. If the alarm is disabled, the system cannot process alarms for that tag.<br>For the PC-Based alarm, the condition for this is, for example, IA > 80; if the tag value for IA is > 80, the appearance will show. The tag is constantly scanned, so the condition triggers the alarm once it is enabled.<br>For the Onboard alarm, the condition for this is a digital tag, which is set by the driver when a new alarm record on the device is read. If the alarm was disabled, the driver cannot set the digital tag. When the alarm is enabled, nothing happens because the alarm was already "processed" by the driver and will never get reprocessed.<br>Thus, there is no resolution. | |
| The number of alarms that display is fewer than the limit set by Alarm Summary length parameter. | This happens when the number of alarms exceeds 1000 and the system has multiple clusters. | Use one or more of these procedures:<br>Set alarm filtering in the alarm viewer to reduce the number of alarms that can display.<br>Only support a one-cluster system.<br>If a multiple-cluster system is necessary, display a separate slarm page for each cluster. |
| Cannot filter on categories for alarms. | The new categories do not display in the list when you want to select them. | Use Custom Filter 8 instead. Currently, it is the only means available for adding custom filtering to alarms. For more information, see *Filtering and Freezing Log Views* on page 147. |
| Page Down button causes an empty page to display. | The last alarm was on the previous page. When there are no more alarms, pressing Page Down displays a blank page. | Click Page Up to return to the previous page (and the last alarms for the system). |

## Event/Alarm Log Columns Table

This table lists the column headings that are available for use when viewing the event log and alarm logs. The field name is the name that you select when you add a column. The display name is the default column heading in the log. To change the display name, see *Changing the Display Name in an Alarm or Event Log*.

© 2011 Schneider Electric All Rights Reserved

| Field Name | Default Display Name | Description |
|---|---|---|
| Tag | Tag | alarm tag |
| Name | Description | description of alarm e.g., Sag Vcn or Under Voltage B-C |
| Cluster | Cluster | cluster name to which the alarm belongs |
| Date (OnDate: event log) | Date | date (MMDDYYYY) that alarm annunciated |
| Time (OnTime: event log) | Time | time (HH:MM:SS) that alarm annunciated |
| TimeMS (OnTimeMS: event log) | Time | time (HH:MM:SS:MS) that alarm annunciated |
| OffDate | OffDate | event log only: date (MMDDYYYY) that alarm dropped out |
| OffTIme | OffTime | event log only: time (HH:MM:SS) that alarm dropped out |
| AckDate | AckDate | event log only: date (MMDDYYYY) that the alarm was acknowledged |
| AckTime | AckTime | event log only: time (HH:MM:SS) that the alarm was acknowledged |
| AckTimeMS | AckTimeMS | event log only: time (HH:MM:SS:MS) that the alarm was acknowledged, with milliseconds |
| State | State | state of the alarm: disappearance, appearance defaults, or customized from the alarm description field |
| LogState | LogState | alarm logs only: active or inactive |
| SumState | State | event log only: state of the entry in the event log: disappearance, appearance |
| AlmComment | AlmComment | event log only: these entries come from the time-stamped digital alarm window “Comment” fields |
| Comment | Comment | event log only: displays comments from the alarm |
| UserName | User | user name associated with the event |
| FullName | FullName | full user name associated with the event |
| custom1 | Equipment | default equipment name displays; used for alarm filtering and viewing |
| custom2 | n/a: waveform information | used for waveform acquisition |
| custom3 | n/a: metadata, categorization | These are all alarm/event attributes. |
| custom4 | n/a: metadata, alarm type | |
| custom5 | n/a: metadata, alarm group | |
| custom6 | n/a: metadata, subcategorization | |
| custom7 | n/a: metadata, alarm level | |
| custom8 | n/a: user filter | used for customized filtering |

## Changing the Display Name in an Alarm or Event Log

Although you cannot delete column headings in the alarm and event logs, you can add and edit them via the System Parameters. To do this:

1. Open your project.
2. Open the Project Editor. Click System > Parameters.


© 2011 Schneider Electric All Rights Reserved

3. Scroll down through the options (do not use the drop down list, use the scroll bar on the right side) until the correct Section Name and Name display. In this documentation, the notation used is:

   [Section Name]Name

   **For the Event Log**, Find record [Format]Summary. If it does not exist, add a new record in the form, and enter the details. The Value field lists all of the columns in the format of:

   {F1,W1}{F2,W2}{F3,W3}... where X1 is the field name (selected when adding columns to a log) for the first column, X2 is the field name for the second column, etc; and W1 is the width of the first column, W2 is the width of the second column, etc. The width specification is optional. The column is automatically set to an initial szie when the page is displayed.

   **For all other alarm pages: Alarm Log, Unacknowledged Alarms Logs, and Disabled Alarms**, use [Format]Alarm. Again, use the Value line to change elements.

   **For equipment pop-up pages**, use [Format]Equipment.

   **For the alarm banner**, use [Format]LastAlarm. To change the display name of a column, replace or add parameter [AlarmHeading]FieldName where FieldName is the field name used in the [Format] parameters. Set the Value field to the display name for the field.
4. After making any changes, compile and run the project. The headings will be changed when you view the logs from the Graphics Page.

## Filtering and Freezing Log Views

### Filtering Information

To filter the information that displays in any of the alarm logs or the event log, click Filter (in the upper left corner of the screen). The Advanced Alarm Filter screen displays. The table below describes its settings.

| Filter Option | Description: Display all alarms for: |
|---|---|
| Basic Filter box: | |
| Start Date/End Date | a date range.<br>Choosing only a start date displays alarms from that date to the current date.<br>Choosing only an end date displays alarms for the past year up to that date.<br>For example, to display alarms only for today's date, enter only a start date. |
| Start Time/End Time | a time range.<br>Choosing only a beginning time displays alarms from that time through the end of the day (23:59:59 or 11:59:59 p.m.).<br>Choosing only an ending time from the start of the day (00:00:00 or 12:00:00 a.m.) through the time selected. |
| Tag | a single tag; use tag name only, do not include equipment name. For example, enter *MMXU1\A\phsA*, not *MainCM4\MMXU1\A\phsA*.<br>To filter on tag and equipment, enter the tag here and the equipment in the Equipment Name field. |
| Equipment Name | a device (entered when using the Profile Wizard or Automation Interface; (listed in Citect Explorer > System > Equipment) |
| Cluster | a single cluster, which was added when setting up the project (listed in Project Editor > Servers > Clusters) |
| Alarm Name | the descriptive information in the Alarm Name field of the Time Stamped Digital Alarms page |

© 2011 Schneider Electric All Rights Reserved

| Filter Option | Description: Display all alarms for: |
|---|---|
| Alarm Description | Alarm Desc from Time Stamped Digital Alarms: a customized on and off text description, such as “active” and “inactive” |
| Custom Filter 8 | There are eight custom filters, which can be assigned by the customer in each alarm. A group of alarms in a specific location could have the same name in CUSTOM8 so that custom filtering can be easily applied.<br>Custom8 has a default assignment of “Equipment.” To change custom filter assignments, use the AlarmFormat parameter (Project Editor > System > Parameters). This is the only means available for filtering on a custom field. When viewing the log, you can use the new custom filter by typing it into the Custom Filter field. |
| Group Filter box: | |
| Categorization | These “alarm filters” are created in the Profile Editor when alarms are created. |
| Alarm Type | |
| Alarm Group | |
| Subcategorization | |
| Alarm Level | |
| Type Filter box: These are advanced PowerLogic SCADA topics; see PowerLogic SCADA help for more information. | |
| State | the state of the alarm |
| Type | the data type established for the alarm tag |
| Area | the area associated with the alarm |
| Category | This is the alarm category. There are four predefined categories (high, medium, low, and event).You can assign alarms to their own categories by changing the equipment profiles and then re-generating the database.<br>See the following table (Categories and Priorities) for a list of the categories and their defaults.<br>Keep in mind that alarms that are categorized as events need to keep the category of _PLS_ ALM_EVENT (category 1004). |
| Priority | This is the priority of the alarm category; not used in the default PLS_Include project.<br>As with the category, priority has defaults (see Categories and Priorities table below). You can change these settings in the equipment profiles. *However, be sure that you use priority 1 for events*. |

| Category Label | Category Number | Priority Number |
|---|---|---|
| _PLSALM_HIGH | 1001 | 1 |
| _PLSALM_MEDIUM | 1002 | 2 |
| _PLSALM_LOW | 1003 | 3 |
| _PLSALM_EVENT | 1004 | 0 |

### Freezing the View

To freeze the view in an alarm window, click Freeze in the upper left corner of the screen. This “locks” the alarm page so that only the alarms that are currently displayed will remain in view. The button name changes to “Unfreeze.” Alarms may continue to occur, but they will not display until you click “Unfreeze.”


© 2011 Schneider Electric All Rights Reserved

## Using the Analysis Page

The Analysis Page offers two options for viewing data trends. In either option, you must select the tag(s) that are to be included. To be available for viewing in trends, a tag must be included in a device profile, and it must have the “Trend Tag” box checked (see *Adding, Editing, or Deleting a Device Profile* on page 52 for more information).

Trend data is automatically logged when you check “Trend Tag” for tag and then add it to the PowerLogic SCADA project. If too many tags are chosen as trend tags, it could cause the hard drive to fill up.

NOTE: The maximum number of tags (pens) that will display correctly on the screen is ten. If you exceed ten pens, labels for these pens will not display correctly. Use one of these methods to correct this issue:

1. Enlarge the window to accommodate the extra pens/labels.
2. Write custom code to cause the labels to always be in the same position, overlapping each other when the trend pen is created. The user can then move the label around for better viewing.
3. As with option number two, control the label positions with code; but then, move the labels back to that same spot when a user selects the trend pen again.

There are two methods of calculating disk space usage: scaled and floating point. For more information on these calculations, see Calculating Disk Storage in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).

- Trending: Use this option to view historical trends. To select tags, click the Add Pen button on the toolbar:

  Then associate the pen with a tag. By default, most trend data is polled every 15 minutes, and it is stored for one year (Citect Project Editor > Tags > Trend Tags) or until it is FIFO’d out. Some tags are polled every 5 seconds and are stored for two weeks. These tags are:

  Current A
  Current B
  Current C
  Apparent Power Total
  Reactive Power Total
  Real Power Total
  Voltage A-B
  Voltage B-C
  Voltage C-A
  Frequency
  Power Factor Total

- Instant Trend: Use this option to view real-time trends. This allows viewing of data that is not set up for storage. To select tags for this trend, click the Instant Trend Selector on the toolbar:

  NOTE: If one of the pens returns a value of "1.#R,' the tag selected was not valid; no number could be reported for it. None of the values for any of the pens in the trend will be updated. To solve this issue, close the trend and open it again. This time, do not include the pen that gave the invalid return.

In either trending option, click Help for help using the tool:

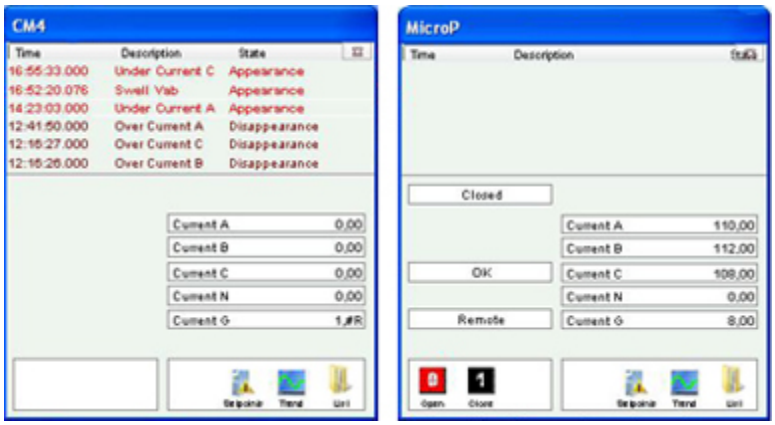
© 2011 Schneider Electric All Rights Reserved

## Using the Equipment Pop-Up Page

See also: *Viewing the Alarms/Events Page* on page 142

This page displays when you click on a device symbol/genie on a single-line page. This page shows a detailed status for a particular device. Some controls on this page are available only to users with certain privilege levels (see *Adding and Modifying User Accounts* on page 160 for user access levels).

One of two status pages displays. The page on the left illustrates the status page for a meter genie. The page on the right illustrates the status page for a circuit breaker genie.




At the top of the page, the most recent alarms and events are listed (racked in/out, Comms Loss, etc.). To view details about an individual alarm or event, right-click the alarm. You can view:

- a waveform. (If you do not see “Waveform” in the list when you right-click the alarm, there are no waveforms for this alarm.) Waveforms can display only if the device is set to “acquire on event,” and the waveform option is checked in the Profile Editor (see *Enabling Waveforms* on page 51).

  When the waveform is available for viewing, it displays when you click this link. Waveforms must be correctly set up before they will display. If there are multiple waveforms, you must select from the list that displays (by default, the waveform search returns all waveform files acquired within the 24 hours prior to the time of the alarm). See *Enabling Waveforms for Onboard Alarms* on page 126 for more information. See *Viewing Waveforms* on page 151 for instructions on viewing waveforms.

  When you select the waveform option, and no waveforms are returned, one of two things is likely:

  — the alarm has been acquired at the device, but it has not yet been passed to PowerLogic SCADA

  — the device was not set to acquire a waveform, and the waveform option was checked in the Profile Editor
- details about the device (currents, voltages, powers, resets, others.),
- Finally, you can acknowledge or disable the alarm. Acknowledged and disabled alarms are moved to their own sub-tabs.

On the left side of the of the status page, status messages display, based on the tags defined for equipment referenced in this genie. The list varies, depending on the device. Possible tags are:

XCBR1\Pos Position (circuit breakers only)


© 2011 Schneider Electric All Rights Reserved

XCBR1\CCBRkdPos Racked Out (circuit breakers only)

XCBR1\CBRkdPos Matching Fault/Trip Circuit Supervision (circuit breakers only)

XCBR1\Loc Local/Remote (circuit breakers only)

XCBR1\ESwPos Earth Switch (circuit breakers only)

PTRC1\Op Tripped

LPHD1\EEHealth Communication Failure

NOTE: For MicroLogic Type P devices, circuit breaker status fields will display #COM if the device does not have a CCM. Thus, you should not add any tags that refer to the CCM, such as Racked In/Racked Out.

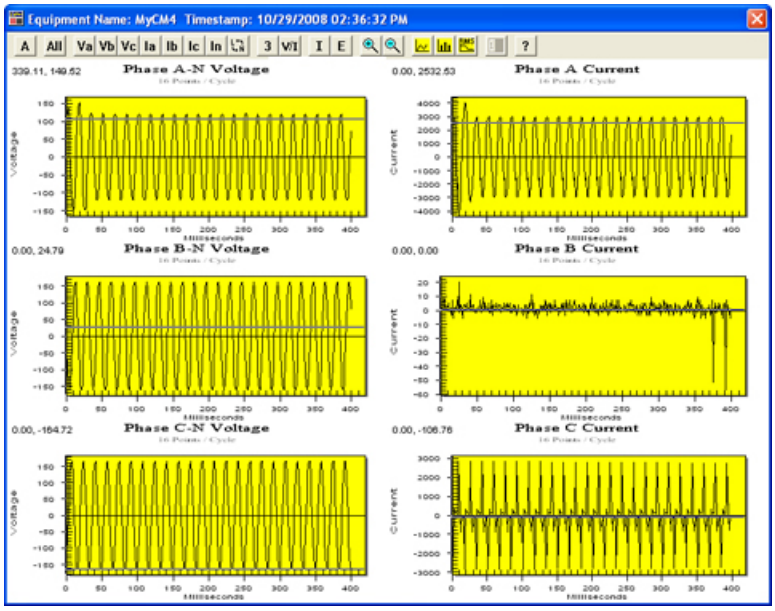
On the right side of the page, real-time values will display for the tag type that you chose in the Value field when you added the genie in the design-time mode. For example, if you enter MMXU1\A\phsA as the value, you will see real-time currents here, as illustrated above. If you did not enter anything in the Value field when adding the genie, this area will be blank.

At the bottom left corner of the circuit breaker status page, Open, Close, Reset (for circuit breakers).

At the bottom right corner, are the Setpoints, Trend, and List options. See the following sections for descriptions.

### Viewing Waveforms

After you select a waveform for viewing from the genie status page (and after the waveform has been sent to PowerLogic SCADA), the waveform displays, as shown below.




The toolbar at the top of the screen includes these buttons:

| Button | Description |
|---|---|
| A | Not currently used. |

© 2011 Schneider Electric All Rights Reserved

| Button | Description |
|---|---|
| All | Click to display all graphs (all three voltages and all three currents of the waveform capture. |
| Va Vb Vc | Click to display individual voltage phases (Va = A-N, Vb = B-N, Vc = C-N). |
| Ia Ib Ic In Ig | Click to display individual current phases (Ia = phase A, Ib = phase B, Ic = phase C, In = neutral, Ig = calculated ground current). |
| | Click to toggle between line-to-line and line-to-neutral voltage phases. |
| 3 | Click to view all three voltage or all three current phases within the same graph. Phase A, B, and C voltages display together. Phase A defaults to black, B to blue, C to red. In the same way, phase A, B, and C currents display together (A is black, B is blue, and C is red).<br>To change colours, right-click the graph title. Select Control Properties. Go to the Colours tab and make changes. |
| V/I | Click to view each phase, both voltage and current, within a single graph. You see phase A voltage and current in one graph, phase B voltage and current in another, and phase C voltage and current in a third. |
| I | Click to display the Import Waveform dialog. If you have SMS installed on the computer, it defaults to the Comtrade folder. If not, it goes to the root of the C drive, for you to locate the waveform you want to import. Double-click the waveform to view it. |
| E | Click to begin exporting a waveform. Select the type of file you will save (bitmap, text, or metafile). Then select the location where you want to store the waveform you are viewing. |
| | Click to zoom in or out. When zoomed in, there is a scroll bar at the bottom of each graph, used to move to the left or right. |
| | Click to view the display as a waveform plot. |
| | Click to view the display as a spectral plot. This feature does not work for the "3" or "V/I" views. |
| | Click to view the display as an RMS plot. |
| | Select any phase; then click here to view the digital status of any of the digital channel(s) that have been set up for this circuit monitor. |
| ? | Not currently used. |

### Entering Setpoints for Alarms

NOTE: Any time you change setpoints, you should immediately restart the project. Otherwise, setpoints will not be properly read (they will be truncated and either rounded down or up to a whole integer).

To add setpoints for alarms, follow these steps:

1. View the page, then click the genie for which you want to enter setpoints. A status window displays with the name of the genie.
2. Click Setpoints, then choose Analog, Digital, or All. When the Alarm Setpoints screen displays, select the first value you want to change. At the “keypad” screen (see below), enter the new value. Click OK to save it. Do this for each setpoint that you want to change.


© 2011 Schneider Electric All Rights Reserved


Based on these setpoints, alarms can begin to display both in the alarms window at the top of the runtime screen and on the Alarms/Events tab (assuming you have set one up for this installation).

When there is a comms loss for a device: the last state before the loss happened is displayed on the screen.

**The indication of loss of communications does not filter through the entire bus animation: the downstream part of the drawing may still appear as if communication is working. When any part of a one-line drawing loses communication, do not continue to trust downstream readings until you address the loss of communication.**

### Viewing Real-Time Trends

This option displays an historical trend. The data that displays is determined by the value that was selected in the Value Type field when this genie was added to the one-line page.

To view a trend: From the one-line page in the runtime environment, click a genie to view its status window. Click Trend, in the lower right corner. The Analyst screen displays for that trend.

You can select the timeframe for the trend. You can also uncheck phases to remove them from the trend, or highlight a phase to bring it to the front of the trend. For detailed information about the buttons on the screen, click “?” at the top of the page.

### Viewing Lists of Real-Time Information for the Genie

To view lists of real-time currents, voltages; powers; resets and controls; and miscellaneous readings, click List, in the lower right corner, then click an item from the list: Currents, Voltages, Powers, Resets, or Others.

For resets and controls, which are interactive, you should assign users a high level of security. For a list of the default user levels, see *Adding and Modifying User Accounts* on page 160. That link also includes information on creating unique users.

When you click an item from the list, individual tag readings display for that tag type (depending on the tags that you have chosen for this device type). When you click any item in that list, the tag pop-up menu displays with these options: Trend, Override Tag, Control Inhibit Tag, and Tag Status. **See Overriding Tag Status**, below, for details.

### Overriding Tag Status

From the list, you can right-click individual tags and override status settings. To access this feature, the user account must be at least level 4.

**Trend**: This link allows you to view a trend for the tag that you clicked.

**Override Tag**: You can use this feature to override a real-time value that is incorrect, or to test graphics. Enter the value that you want the system to "read" for this tag in the Override Value line. When you click Apply, the tag is highlighted. When you have finished the test, return to this list to remove the override.

© 2011 Schneider Electric All Rights Reserved

**Control Inhibit Tag**: When this feature is ON, you will not be able to process writes for this tag. To enable this inhibit, click Apply for this tag from the list. The tag reading is highlighted. To disable this feature, return to the list view of this tag; click Remove.

You can perform control inhibit on an entire device. To do this, you will use the IODeviceControl Cicode function. For more inforrmation, see the I/O Device Properties topic in the Citect. chm help file (I/O Devices Properties).

**Tag Status**: This screen views the status of the display value, override status, control inhibit status, and field value. You can also change the override status and control inhibit status on this screen.

**Changing background colours**: Default colours are assigned for for the tag override and control inhibit. Change the default background colours in the parameters, not in the ini file.

To change the colour for tag overrides, use OverrideTextBackgroundColor. To change the colour for control inhibits, use ControlInhibitTextBackgroundColor. For detailed help, see Page Parameters in the Parameters.chm help.


© 2011 Schneider Electric All Rights Reserved

# Multi-Monitor Support

PowerLogic SCADA provides the means for viewing the runtime environment from multiple monitors. This setup allows you to view unrelated information at separate monitors that are placed beside each other. In this example, there are four monitors, configured left to right in a single row:

**Main Monitor Monitor 2 Monitor 3 Monitor 4**

(displays Windows Start button)

In the Citect.ini file for the project (Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerLogic SCADA 7.20\User\[projectname] or Vista: ProgramData\Schneider Electric\PowerLogic SCADA 7.20\User\[projectname]), add the following instructions:

Add the following instructions to the Citect.ini file (changes can be added directly to the Citect.ini file or in the project database, via menu command: System > Parameters in Project Editor):

[MultiMonitors]

Monitors = 4

StartupPage1 =

StartupPage2 =

StartupPage3 =

StartupPage4 =

where

Monitors = n; n = number of monitors used to display the pages in the project

PowerLogic SCADA displays the windows according to the formation of the monitors—such as left-to-right, top-to-bottom or grid—set in the Windows operating system. By default, this is set via the Windows Start menu > Control Panel > Display applet.

StartupPage1 = startup page that needs to be displayed on monitor 1

StartupPage2 = startup page that needs to be displayed on monitor 2

StartupPage3 = startup page that needs to be displayed on monitor 3

StartupPage4 = startup page that needs to be displayed on monitor 4

© 2011 Schneider Electric All Rights Reserved

# Using the Graphics Test Mode

Graphics Test Mode is to be used for testing or demonstration during system setup, prior to commissioning. Always disable Graphics Test Mode before you release the product to the end user.

Graphics Test Mode can be enabled by setting the following parameter in the project parameters:

Section Name: PowerLogicSCADA

Name: GraphicsTest

Value: Enable Graphics Test Mode

To disable, remove this parameter information.

Graphics Test Mode is designed to enable off-line testing of animation in single-line drawings. This feature is useful for project development, prior to commissioning a system. Do not use this mode with physical devices that are connected to a live environment.

NOTE: This does not prevent control to a device. The default graphics. when in Graphics Test Mode, will write only to status tags for open, close, and breaker reset. If you edit these graphics, or create new ones, you can write to control tags even when you are in the Graphics Test Mode.

There are generic devices available for Graphics Test Mode. These devices have this naming convention:

**Breaker_1** through **Breaker_50**

and

**Meter_1** through **Meter_5**.


© 2011 Schneider Electric All Rights Reserved

# Using the Demo Mode

You can run PowerLogic SCADA without the hardware key in demonstration (Demo) mode. In demonstration mode, you can use all PowerLogic SCADA features normally, but with restricted runtime and I/O.

If you configure PowerLogic SCADA to run as multiple processes on one CPU or multiple CPUs, you cannot use PowerLogic SCADA in demo mode; but if you run PowerLogic SCADA as one process, you can use a demo mode. The following demonstration modes are available:

- 15 minutes with a maximum of 50,000 real I/O.
- 10 hours with no static points and a maximum of one dynamic real I/O. This is useful for demonstrations using memory and disk I/O. PowerLogic SCADA starts in this mode if no static points are configured.

  If you want to demonstrate DDE, CTAPI, or ODBC writes to PowerLogic SCADA in this mode, you can only write one point. To write to more than one point, you must force PowerLogic SCADA to start in 15 minute-50,000 I/O demo mode by creating at least one static I/O point.

  For this to work, you must configure a real variable tag, with an accompanying PLC or I/O device. The tag must be used by a page or in Cicode. If you do not have a real I/O device connected, PowerLogic SCADA displays a hardware error, which you can disable using the IODeviceControl function.
- 8 hours with a maximum of 42,000 real I/O. This is only available through special PowerLogic SCADA Integration Partners (CIP) keys.

© 2011 Schneider Electric All Rights Reserved

# Assigning and Controlling User Privileges

NOTE: You need to give users appropriate levels of access, depending on the work they will do. For safety reasons, only advanced users should be given access to such features as controls and resets.

This chapter discusses how to establish user security for the system. This section includes the following topics:




© 2011 Schneider Electric All Rights Reserved

# Default User Access Settings (Privileges)

User access rights (privileges) are defined in System > Roles form, in the Project Editor. The following table describes the access rights. These privileges are included in PLSSecurity.ci file (in the PLS_Include project). Any changes made to this file will be overwritten when PLS_Include is updated (every new release). You need to document every change you make, so that you can update PLSSecurity.ci when PLS_Include is upgraded.

| Access Right | | | Roles / Global Privileges | | | | |
|---|---|---|---|---|---|---|---|
| Description | Access Level Label | Access Level Value | Operator1 | Operator2 | Engineer | Administrator | Kernel |
| | | | 2 | 4 | 6 | 7 | 8 |
| Circuit breaker, switch control | PL_Sec_CBControl | 1 | | X | X | X | X |
| IED configuration | PL_Sec_IEDConfig | 2 | | | X | X | X |
| Circuit breaker tagging | PL_Sec_Tagging | 3 | | | X | X | X |
| Alarms acknowledgment | PL_Sec_AlmAck | 4 | X | X | X | X | X |
| Alarm deletion | PL_Sec_AlmDelete | 5 | | | X | X | X |
| Alarm configuration | PL_Sec_AlmConfig | 6 | | | X | X | X |
| Add/remove log-in users | PL_Sec_UserConfig | 7 | | | | X | X |
| Reset alarms in device | PL_Sec_AlmReset | 8 | | | X | | X |
| Shutdown runtime | PL_Sec_Shutdown | 9 | | | X | X | X |
| View waveforms | PL_Sec_ViewWaveform | 10 | X | X | X | X | X |

NOTE: Privileges 1, 3, and 5 are currently unused. You can use them as you wish. See *Changing Access Rights* on page 162 for instructions on changing access level privileges.

A method of limiting access to the design time/configuration environment is to remove the user rights to certain PowerLogic SCADA files. From the server, use Windows security to remove user accounts from individual features. For example, access to the following list of .exe files should be restricted to users who have design time/configuration privileges.

- CtDraw32.exe - Graphics builder
- CtEdit32.exe - Project Editor
- CtExplor.exe - Project Explorer
- ProfileWizard.exe - Profile Wizard
- ColorSwap.exe - Color Swap Tool
- ProfileEditor.exe - Profile Editor

All of these files need to have the same level of security, as they are interrelated.

Additionally, to prevent users from accessing and changing the code responsible for enforcing user security in the runtime environment, you need to lock down the user rights for the Cicode files (.ci extension) in the PLS_Includes project.

NOTE: Another way to lock users out from changing an existing project is to implement read-only projects.

© 2011 Schneider Electric All Rights Reserved

# Adding and Modifying User Accounts

You must add at least one user to any project before you can run and view it. Each user must have a role and a user account. Terms you need to understand are:

- privilege: the level of access that is applied to a system element; a user account has individual privileges, which the user can then control
- role: contains a defined set of privileges, that are assigned to users

The Users screen controls all of the user access levels for each project. To assign PowerLogic SCADA user access, follow these steps:

1. From the PowerLogic SCADA Explorer, choose the project for which you want to assign user access.
2. From the Project Editor, click System > Roles.
3. For the first user, assign a user role, Windows group name (optional), and global privileges. If you need additional information, click Help from that screen. For global privileges, see the table in *Default User Access Settings (Privileges)* on page 159 for the level of each type of access right.
4. After you add the first role, click Add.
5. From the Project Editor, click System > Users.
6. Assign a user account. If you need additional information, click Help from that screen.
7. After you add the first user, click Add.
8. To add additional users, do the following: with a user displayed, click Add (which creates a copy of the user); type the new user information in place of the old information; then click Replace to overwrite the information.

   Note that the record count increases by one with each addition. To view the users one at a time, scroll through the list.

## Using Windows Integrated Users

Instead of using the above method to define users in the project, you can incorporate PowerLogic SCADA users and security options with the standard Windows security system. Using the integrated Windows security feature, the Windows user can log on to PowerLogic SCADA runtime with runtime privileges and areas configured within the project. For a Windows user to be able to log on to runtime, it must be linked to a PowerLogic SCADA "role," which is defined in the project with associated privileges.

To assign Windows integrated user access,

To link a Windows user to a PowerLogic SCADA role, you should add the "role" that specifies the Windows group of which the Windows user is a member.

The pre-existing AutoLogin capability is extended to include the client, when the user is a Windows user, having an associated PowerLogic SCADA role. In order to invoke this functionality for a Windows user, you need to set the [Client]AutoLoginMode parameter in the Citect.ini file.

Instead of using auto-login when the system starts up, users can also log in to PowerLogic SCADA using any Windows user credential that is a member of the linked group.

When the name of a PowerLogic SCADA user also has the same name as a Windows user, the PowerLogic SCADA user takes priority at runtime. However, if a valid PowerLogic SCADA user login fails for some reason, the Windows user credentials will not be checked and an alert will be generated to advise that the login was not effective.


© 2011 Schneider Electric All Rights Reserved

For more information, see Using PowerLogic SCADA with Windows Security in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).

© 2011 Schneider Electric All Rights Reserved

# Changing Access Rights

You can edit the default access rights to each of the eight levels, thus changing the privileges that are available at each level. This is done in the PLSSecurity.ci file (in the PLS_Include project).

To change access rights:

1. Open PLSSecurity.ci.
2. Locate the AccessRights section.
3. For each right that you want to add (for example, to add access privileges for working with switches), add a new CASE.
4. Save and close PLSSecurity.ci.

NOTE: PLSSecurity.ci is overwritten any time that the PLS_Include project is overwritten. This happens with every release of PowerLogic SCADA, including service packs. To ensure that you do not lose changes that you enter, you need to:

- Note the changes that you make. Keep a copy of these changes
- Re-merge the changes every time that the PLS_Include project is overwritten.


© 2011 Schneider Electric All Rights Reserved

# Appendix 1: Default Genie Library

The genie library includes a number of general genies for objects such as motors and pumps. There are also genies that are specific to PowerLogic. These genies use a particular naming convention, which is described in the table below. In the PowerLogic library, each genie name begins with “pls,” and is followed by a description of the type of genie according to this table:

| first | second | third | fourth |
|---|---|---|---|
| pls<br>indicates PowerLogic library | alarm = alarm | base = primitive genies | 1 = small |
| | ansi = ANSI style | cb = circuit breaker | 2 = large |
| | display = equipment details | sw = switchgear | |
| | gen = generic | cmd = control genies | |
| | iec = IEC style | eq = equipment (devices) | |
| | style = navigation | | |
| | | | |

Additionally, the actual genies have abbreviated names. When you highlight a genie name, the abbreviation displays above the top row of genie icons.

The following tables list and define the individual genies in each of the PowerLogic libraries.

### PLS_ALARM

This library includes genies that provide functionality to alarm displays.

| Genie Abbreviation | Description |
|---|---|
| Field | data portion of an alarm row |
| Row | a genie with a collection of fields |
| Selector | for column resizing |
| Setpoint | a setpoint row |

### PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2

These libraries include a variety of base symbols used to created genies for ANSI equipment.

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| sl_battery_gen | single-cell battery |
| sl_battery_multi | multiple-cell battery |
| sl_capacitor | capacitor |
| sl_capacitor_vari | variable capacitor |
| SL_Closed_HV | closed circuit breaker position |
| sl_closed_knife | closed knife switch |
| sl_closed_lv | closed low-voltage circuit breaker |
| sl_conductive_path_1 | conductive path 1 |

© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| sl_conductive_path_2 | conductive path 2 |
| sl_conductive_path_3 | conductive path 3 |
| sl_conductor_junction | conductor junction |
| sl_contact_nc_closed | contact break, closed |
| sl_contact_nc_open | contact break, open |
| sl_contact_no_closed | contact make, closed |
| sl_contact_no_open | contact make, open |
| sl_contact_term | contact terminal |
| sl_ct | current transformer |
| sl_in_cb_rack | incoming, circuit breaker racked out, plug |
| SL_In_Rack | incoming, circuit breaker racked out, socket |
| sl_in_sw_head | incoming switch head |
| sl_inductor | inductor |
| sl_inductor_adjust | adjustable inductor |
| sl_inductor_gen | general inductor |
| sl_inductor_magcore | magnetic core inductor |
| sl_inductor_vari | variable inductor |
| sl_open | open symbol |
| sl_open_fuse_sw | open isolating fuse-switch |
| SL_Open_HV | open high-voltage circuit breaker |
| sl_open_knife | open knife-type switch |
| SL_Open_LV | open low-voltage circuit breaker |
| sl_out_cb_rack | outgoing, circuit breaker racked out, plug |
| SL_Out_Rack | outgoing, circuit breaker racked out, socket |
| sl_pb_break | push-button, break |
| sl_pb_make | push-button, make |
| sl_pb_term | push-button, terminal |
| sl_pt | potential transformer |
| sl_relay | relay |
| sl_resistor | resistor |
| sl_resistor_adjust | adjustable resistor |
| sl_resistor_vari | variable resistor |
| sl_separable_con_closed | separable connector, closed |
| sl_separable_con_open | separable connector, open |
| sl_separable_con_plug | separable connector plug |
| sl_separable_con_socket | separable connector socket |

### PLS_ANSI_CB_1 / PLS_ANSI_CB_2

These libraries include genies for ANSI-type high-voltage and low-voltage drawout circuit breakers.

1 = small size

2 = large size


© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| hv_cb_bus_dr_c | high-voltage drawout circuit breaker, remote control |
| hv_cb_bus_dr_nc | high-voltage drawout circuit breaker, not remote |
| hv_cb_bus_nd_c | high-voltage non-drawout circuit breaker, not remote |
| hv_cb_bus_nd_nc | high-voltage non-drawout circuit breaker, not remote |
| hv_cb_fd_dr_c | high-voltage, drawout circuit breaker, remote control, with earth at bottom |
| hv_cb_fd_dr_nc | high-voltage, drawout circuit breaker, no remote control, with earth at bottom |
| hv_cb_fd_nd_c | high-voltage, non-drawout circuit breaker, remote control, with earth at bottom |
| hv_cb_fd_nd_nc | high-voltage, non-drawout circuit breaker, no remote control, with earth at bottom |
| hv_cb_inc_dr_c | high-voltage, drawout circuit breaker, remote control, with earth at top |
| hv_cb_inc_dr_nc | high-voltage, drawout circuit breaker, no remote control, with earth at top |
| hv_cb_inc_nd_c | high-voltage, non-drawout circuit breaker, remote control, with earth at top |
| hv_cb_inc_nd_nc | high-voltage, non-drawout circuit breaker, no remote control, with earth at top |
| hv_cb_nes_dr_c | high-voltage drawout circuit breaker, remote control, no earth |
| hv_cb_nes_dr_nc | high-voltage drawout circuit breaker, no remote control, no earth |
| hv_cb_nes_nd_c | high-voltage non-drawout circuit breaker, remote control, no earth |
| hv_cb_nes_nd_nc | high-voltage non-drawout circuit breaker, no remote control, no earth |
| lv_cb_bus_dr_c | low-voltage drawout circuit breaker, remote control, busbar-type with earth at bottom |
| lv_cb_bus_dr_nc | low-voltage drawout circuit breaker, no remote control, busbar-type with earth at bottom |
| lv_cb_bus_nd_c | low-voltage non-drawout circuit breaker, remote control, busbar-type with earth at bottom |
| lv_cb_bus_nd_nc | low-voltage non-drawout circuit breaker, no remote control, busbar-type with earth at bottom |
| lv_cb_fd_dr_c | low-voltage drawout circuit breaker, remote control, earth on load side (bottom of drawing) |
| lv_cb_fd_dr_nc | low-voltage drawout circuit breaker, no remote control, earth on load side (bottom of drawing) |
| lv_cb_fd_nd_c | low-voltage non-drawout circuit breaker, remote control, earth on load side (bottom of drawing) |
| lv_cb_fd_nd_nc | low-voltage non-drawout circuit breaker, no remote control, earth on load side (bottom of drawing) |
| lv_cb_inc_dr_c | low-voltage drawout circuit breaker, remote control, earth on feeder (top of drawing) |
| lv_cb_inc_dr_nc | low-voltage drawout circuit breaker, no remote control, earth on feeder (top of drawing) |
| lv_cb_inc_nd_c | low-voltage non-drawout circuit breaker, remote control, earth on feeder (top of drawing) |
| lv_cb_inc_nd_nc | low-voltage non-drawout circuit breaker, no remote control, earth on feeder (top of drawing) |
| lv_cb_nes_dr_c | low voltage drawout circuit breaker, no earth, remote control |
| lv_cb_nes_dr_nc | low voltage drawout circuit breaker, no earth, no remote control |
| lv_cb_nes_nd_c | low voltage non-drawout circuit breaker, no earth, remote control |
| lb_cb_nes_nd_nc | low voltage non-drawout circuit breaker, no earth, no remote control |

© 2011 Schneider Electric All Rights Reserved

### PLS_ANSI_SW_1 / PLS_ANSI_SW_2

These libraries include ANSI-style switches:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| sw_fused | switch: feeder, fused |
| sw_fused_isolated | switch: feeder, fused, isolated |
| sw_general | switch: feeder, general |
| sw_knife | switch: knife type |

### PLS_DISPLAY

This library includes two genies that provide data row items for equipment.

| Genie Abbreviation | Description |
|---|---|
| equiplistitem | data row for the equipment tag list |
| EquipValueItem | data row for the equipment popup |

### PLS_GEN_BASE_1 / PLS_GEN_BASE_2

These libraries include a variety of "parts" related to generators, motors, and transformers.

| Genie Abbreviation | Description |
|---|---|
| chassis_ground | chassis ground |
| Dev_Base | device base |
| es_inc | earth switch, incomer |
| es_out | earth switch, feeder |
| Gen_1 | generator, option 1 |
| Gen_2 | generator, option 2 |
| gen_AC | generator: AC |
| gen_DC | generator: DC |
| genset | engine-generator |
| ground | ground |
| Motor_1 | motor, option 1 |
| Motor_2 | motor, option 2 |
| motor_ac | motor: AC |
| motor_dc | motor: DC |
| motor_synch | motor: synchronous |
| SL_Base | circuit breaker base symbol |
| sl_br_in | circuit breaker line in, non-drawout |
| sl_br_out | circuit breaker line out, non-drawout |
| SL_Bustie | bus tie |
| SL_CommLoss | comms loss |
| SL_Discrepancy | position discrepancy |


© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| sl_harmonic_filter_1 | harmonic filter 1 |
| sl_harmonic_filter_2 | harmonic filter 2 |
| SL_In | incoming bus |
| SL_Local | local, rather than remote control |
| SL_Out | feeder |
| SL_Tripped | tripped |
| Test_CB_Control | health test for the circuit breaker control |
| transformer_1_in | transformer 1: general, on-line |
| transformer_1_in_y | transformer 1: star (wye), on-line |
| transformer_1_out | transformer 1: general, off-line |
| transformer_1_out_d | transformer 1: delta, off-line |
| transformer_1_out_y | transformer 1: star (wye), off-line |
| transformer_2_in | transformer 2: general, on-line |
| transformer_2_in_Y | transformer 2: star (wye), on-line |
| transformer_2_out | transformer 2: general, off-line |
| transformer_2_out_D | transformer 2: delta, off-line |
| transformer_2_out_Y | transformer 2: star (wye), off-line (no 2 IN D? or 1 IN D?) |

### PLS_GEN_CMD_1 / PLS_GEN_CMD_2

These libraries include genies that control display of popups and values:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| CmdDetail | provides access to the equipment detail popup |
| cmddetail_meter | provides access to the meter detail popup |
| Control | control in a circuit breaker |
| value | value section of a circuit breaker |
| value_meter | value section of a meter |

### PLS_GEN_EQ_1 / PLS_GEN_EQ_2

These libraries include the general equipment used to make up generators, motors, and transformers:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| busbar_horz | horizontal busbar |
| busbar_vert | vertical busbar |
| gen_ac | generator: AC |
| gen_dc | generator: DC |
| gen_nd_1 | generator 1: no current designation |

© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| gen_nd_2 | generator 2: no current designation |
| mot_ac | motor: AC |
| mot_dc | motor: DC |
| mot_nd_1 | motor 1: no current designation |
| mot_nd_2 | motor 2: no current designation |
| mot_syn | motor, synchronous |
| trans_nd_1 | transformer 1: no connection designation |
| trans_nd_2 | transformer 2: no connection designation |
| trans_sd_1 | transformer 1: star-delta (wye-delta) |
| trans_sd_2 | transformer 2: star delta (wye-delta) |
| trans_ss_1 | transformer 1: star-star (wye-wye) |
| trans_ss_2 | transformer 2: star-star (wye-wye) |

### PLS_IEC_BASE_1 / PLS_IEC_BASE_2

These libraries include a variety of symbols for IEC equipment:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| sl_cap_bank_tuned_3 | capacitor bank 3: tuned |
| sl_cap_bank_tuned_4 | capacitor bank 4: tuned |
| sl_capacitor | capacitor |
| sl_capacitor_vari | capacitor, variable |
| sl_closed | closed switch |
| sl_contact_nc | contact break |
| sl_ct | contact |
| sl_fuse_1 | fuse, option 1 |
| sl_fuse_2 | fuse, option 2 |
| SL_Head | head |
| sl_head_2 | head |
| sl_in_cb_rack | incoming, circuit breaker when racked out, plug |
| SL_In_Rack | incoming, circuit breaker when racked out, socket |
| sl_in_sw_hd_isol | incoming, switch head, isolated |
| sl_in_sw_head | incoming, switch head |
| sl_inductor | inductor |
| sl_inductor_adjust | inductor, adjustable |
| SL_Open | open |
| sl_out_cb_rack | feeder, circuit breaker when racked out, plug |
| SL_Out_Rack | feeder, circuit breaker when racked out, socket |
| sl_resistor | resistor |
| sl_resistor_adjust | resistor with adjustable contact |
| sl_resistor_vari | resistor, variable |


© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| sl_sw_static_1 | static switch 1 |
| sl_sw_static_2 | static switch 2 |

### PLS_IEC_CB_1 / PLS_IEC_CB_2

These libraries include high-voltage drawout circuit breakers:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| hv_cb_bus_dr_c | high-voltage drawout circuit breaker, remote control |
| hv_cb_bus_dr_nc | high-voltage drawout circuit breaker, no remote control |
| hv_cb_bus_nd_c | high-voltage non-drawout circuit breaker, remote control |
| hv_cb_bus_nd_nc | high-voltage non-drawout circuit breaker, no remote control |
| hv_cb_fd_dr_c | high-voltage, drawout circuit breaker, remote control, earth at bottom |
| hv_cb_fd_dr_nc | high-voltage, drawout circuit breaker, no remote control, earth at bottom |
| hv_cb_fd_nd_c | high-voltage, non-drawout circuit breaker, remote control, earth at bottom |
| hv_cb_fd_nd_nc | high-voltage, non-drawout circuit breaker, no remote control, earth at bottom |
| hv_cb_inc_dr_c | high-voltage, drawout circuit breaker, remote control, earth at top |
| hv_cb_inc_dr_nc | high-voltage, drawout circuit breaker, no remote control, earth at top |
| hv_cb_inc_nd_c | high-voltage, non-drawout circuit breaker, remote control, earth at top |
| hv_cb_inc_nd_nc | high-voltage, non-drawout circuit breaker, no remote control, earth at top |
| hv_cb_nes_dr_c | high-voltage drawout circuit breaker, remote control, no earth |
| hv_cb_nes_dr_nc | high-voltage drawout circuit breaker, no remote control, no earth |
| hv_cb_nes_nd_c | high-voltage non-drawout circuit breaker, remote control, no earth |
| hv_cb_nes_nd_nc | high-voltage non-drawout circuit breaker, no remote control, no earth |

### PLS_IEC_SW_1 / PLS_IEC_SW_2

These libraries include IEC-style switches:

1 = small size

2 = large size

| Genie Abbreviation | Description |
|---|---|
| sw_general | general switch |
| sw_isolated | isolated switch |

### PLS_METER

This library includes meter symbols.

| Genie Abbreviation | Description |
|---|---|
| circuit monitor | PowerLogic SCADA circuit monitor |
| egx | PowerLogic SCADA EGX |
| generic_meter | generic meter |

© 2011 Schneider Electric All Rights Reserved

| Genie Abbreviation | Description |
|---|---|
| ion_7650 | ION 7650 meter |
| micrologic | all Mircologic meters |
| power_meter | PowerLogic SCADA power meter |
| quantum | PowerLogic SCADA Quantum |
| sepam | all Sepam metersI |

### ITEM1

This library includes miscellaneous symbols.

| Genie Abbreviation | Description |
|---|---|
| Item1 | value type and units block for a circuit breaker |
| Item2 | value type and units block for a circuit breaker |
| tab1 | menu tab |
| Tab2 | menu tab |


© 2011 Schneider Electric All Rights Reserved

# Appendix 2: Using Citect INI Parameters

There are a number of Citect INI parameters that you may use to configure driver parameters. These settings may be configured at the protocol level, cluster level, port level, or device level. More specific settings will override a general one. The order of precedence is:

Protocol Name > Cluster Name > Port Name > I/O Device Name

The level at which you want the INI settings to be in effect determines the name you define. For example:

To set the default timeout for all devices using the Micrologic protocol, use:

**[MICROLOGIC]**

**Timeout = 2000**

To override this default for cluster ‘Cluster_1,’ use:

**[MICROLOGIC.Cluster_1]**

**Timeout = 1000**

To override the default value for port ‘Port_1’ on cluster ‘Cluster_1,’ use:

**[MICROLOGIC.Cluster_1.Port_1]**

**Timeout = 3000**

To override the default value for I/O device ‘CircuitBreaker_1’ on port ‘Port_1’ on cluster ‘Cluster_1,’ use:

**[MICROLOGIC.Cluster_1.Port_1.CircuitBreaker_1]**

**Timeout = 4000**

Most settings can be configured to be specific to a particular I/O device. Exceptions are noted in the description for the individual parameter.

## Parameters Database

All INI parameters described in the sections below can be set in the Parameters database. Using special syntax, you can access the parameters in the Project Editor (System < Parameters):

- The section name generally corresponds to the INI section name, although it includes the protocol name, cluster name, and primary device name only.
- The name is the INI value name.

If the parameter is set in the Parameters database, it becomes a new default for either protocol, cluster, or a concrete device (depending on the section name hierarchy).

Examples:

Section Name: [MICROLOGIC.Cluster_1.Breaker_1]

Name: Timeout

Value: 2000

This defines a new default timeout value for a redundant pair of MicroLogic devices (primary device is named Breaker_1 in Cluster_1).

Section Name: [PWRMODBUS.Cluster1]

Name: UseWriteMultiRegistersOnly

Value: 0

This sets UseWriteMultiRegistersOnly to 0 for all PWRMODBUS devices in Cluster 1.

© 2011 Schneider Electric All Rights Reserved

The INI file is read after the parameter database is processed; thus the override options are set in the Parameters database.

In this section, you will find parameters organized into these categories:




© 2011 Schneider Electric All Rights Reserved

# General PowerLogic Driver Parameters

The following parameters are common to all PowerLogic SCADA devices.

## watchtime

Controls how often PowerLogic SCADA will interrogate the driver to determine whether it is still online. This parameter can only be configured for an entire driver, and hence will have the driver dll name as its section name. Where another setting may be [PM870], to set this setting it must be [PLOGIC], as PLOGIC is the name of the dll. This is the only parameter whose section name is defined in this fashion.

Parameter type: seconds

Default Value: 2

Example: [SEPAM] watchtime = 5

## kernelStatisticUpdateRate

Controls how frequently the statistics displayed in the driver kernel window are updated. This time period can be increased in order to decrease CPU load. This parameter can only be configured for the entire protocol (as with the watchtime parameter); it will have the driver dll name as its section name.

Parameter type: milliseconds

Default value: 5000

Examples:

[SEPAM40]

kernelStatisticUpdateRate = 20000

[SEPAM80]

kernelStatisticUpdateRate = 10000

## UseWriteMultiRegistersOnly

Controls PWRMODBUS driver behaviour when a single register is to be written. This parameter is set to 1 by default, enabling all writes to be made using "write multiple registers" MODBUS function. Setting this parameter to 0 allows driver to perform write using "write single register" function if (and only if) one MODBUS register is about to be written in current operation.

Parameter type: integer

Default value: 1

Examples:

[PWRMODBUS]

UseWriteMultiRegistersOnly = 1

[PWRMODBUS.MYCLUSTER.PORT_1.BCM1]

UseWriteMultiRegistersOnly = 0

© 2011 Schneider Electric All Rights Reserved

## timeout

Controls how long the driver waits for a response from a device before setting that device as offline. This value should be greater than the device/gateway timeout period. A timed out request will not be retried. The reason for this is that TCP is a guaranteed transport mechanism, and the lack of a response indicates device failure or communications failure to that device. A device connected via a gateway should use the gateway's retry mechanism.

Parameter type: milliseconds

Default value: 5000

Examples:

[SEPAM40]

Timeout = 2000

[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]

Timeout = 15000

## retry

Defines the number of retry attempts for specific MODBUS requests. Retries may occur either when the request is timed out or certain MODBUS exception reply messages are received. The exact behaviour is controlled by the RetryTimeout and RetryException parameters.

Parameter type: number of attempts

Default value: 3

Examples:

[SEPAM40]

retry = 1

[SEPAM40.MYCLUSTER.PORT_1.SEPAM_DEVICE]

retry = 5

## RetryTimeout

When enabled (by default), the driver will re-try a timed-out MODBUS request.

Parameter type: long (boolean)

Default value: 1

Examples:

[SEPAM40]

RetryTimeout = 1

[SEPAM40.MYCLUSTER.PORT_2.SEPAM_DEVICE]

RetryTimeout = 0


© 2011 Schneider Electric All Rights Reserved

## RetryException

When enabled (disabled by default), the driver will re-try a MODBUS request that has received MODBUS Exception messages. The number of retries is defined by the Retry parameter.

When Retry Exception is enabled, retry occurs when any of the following MODBUS exception messages is received:

SLAVE_DEVICE_FAILURE_EXCEPTION = 0x5

GATEWAY_PATH_UNAVAILABLE_EXCEPTION = 0xA

GATEWAY_TARGET_DEVICE_FAILED_TO_RESPOND_EXCEPTION = 0xB

SLAVE_DEVICE_BUSY_EXCEPTION = 0x6

MEMORY_PARITY_ERROR_EXCEPTION = 0x8

NEGATIVE_ACKNOWLEDGE_EXCPETION = 0x7

Parameter type: long (boolean)

Default value: 0

Examples:

[SEPAM40]

RetryTimeout = 1

[SEPAM40.MYCLUSTER.PORT_2.SEPAM_DEVICE]

RetryTimeout = 0

## standbyRefreshRate

Controls how often a standby IO server attempts to poll a device to update its cache. This time period determines the maximum age that values may be when switching from a primary IO server to a standby. Decreasing this value degrades communications to the device.

Parameter type: seconds

Default value: 60

Examples:

[SEPAM40]

standbyRefreshRate = 30

[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]

standbyRefreshRate = 120

## standbyCheckTime

Controls how often the driver will enquire of PowerLogic SCADA as to whether it is in standby or primary mode. This value can be increased to reduce CPU load.

Parameter type: milliseconds

Default value: 500

Examples:

[SEPAM40]

© 2011 Schneider Electric All Rights Reserved

standbyCheckTime = 500

[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]
standbyCheckTime = 1000

## statusUnitCheckTime

This parameter defines how frequently the driver will try to re-establish the connection with a device that has gone offline on a port that is not disconnected. It sets the maximum rate at which the driver enquires of the device, to determine if it is still operational. If the "watchtime" parameter is set to a longer time, that value will be used instead.

NOTE: If a network gateway has multiple devices connected to it, and one device is disconnected, the driver takes it offline and does not try to reconnect it according to this parameter's schedule. If the port is taken offline and then is reconnected, the driver will reconnect the devices immediately.

Parameter type: seconds

Default value: 5 (20 for MicroLogic)

Example:

[SEPAM40]
statusUnitCheckTime = 5

[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]
standbyCheckTime = 10

## initUnitCheckTime

Controls how long the driver waits before attempting to bring a device online after it has gone offline. This value can be decreased to bring offline devices back into service in a shorter period of time. In a multi-drop scenario, this time should be relatively long, to prevent init unit requests from stalling communications to the rest of the devices on that port.

Parameter type: seconds

Default value: 120

Examples:

[SEPAM40]
initUnitCheckTime = 5

[SEPAM40.MYCLUSTER.PORT_1]
initUnitCheckTime = 120

## initCacheTimeout

Controls how long the driver will spend attempting to populate the cache before bringing a device online. When a tag has been incorrectly configured, the device will come online after this period of time.

Parameter type: seconds

Default value: 60


© 2011 Schneider Electric All Rights Reserved

Examples:

[SEPAM40]

initCacheTimeout = 60

[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]

initCacheTimeout = 30

## cacheRefreshTime

Controls the maximum rate at which the driver will attempt to repopulate its cache. If the driver cannot refresh its cache within the time period specified, it will collect data as fast as the network allows.

Parameter type: milliseconds

Default value: 500

Examples:

[SEPAM40]

cacheRefreshTime = 1000

[SEPAM40.MYCLUSTER.PORT_1.FAST_SEPAM]

cacheRefreshTime = 200

[SEPAM40.MYCLUSTER.PORT_1.UNIMPORTANT_DEVICE]

cacheRefreshTime = 5000

## StatusRegister

Defines a holding register that the driver reads to determine whether a device is responding to communication requests. The result of this read is not important, however it must be a valid register address within the device.

Parameter type: register address

Default value: 1100 or 2 (Sepam)

Examples:

[PWRMODBUS]

statusRegister = 1000

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

statusRegister = 16000

## StatusRegisterType

Used together with StatusRegister; defines the type of the status register. Can only be configured for the PWRMODBUS driver. This parameter can have one of the following values:

0 - HOLDING register (default)

1 - INPUT register

2 - COIL register

3 - DIGITAL input (input coil) register

Any other value equals the default.

© 2011 Schneider Electric All Rights Reserved

Parameter type: register type

Default value: 0

Example:

[PWRMODBUS]

statusRegister = 1000

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

statusRegister = 16000

statusRegisterType = 2

## ModbusBase

Defines the base address for a device. Some MODBUS device registers are defined using a base address of 1. in this case, reading register 100 would actually require reading register 99. In other devices (such as the Sepam) the base address is 0. This parameter allows the base address to be configured according to the device.

Parameter type: integer

Default value: 0 for Sepam; 1 for all other drivers

Examples:

[PWRMODBUS]

ModbusBase = 1

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

ModbusBase = 0

## RegMode

Specifies the order of bytes in a device register. It can only bet set for PWRMODBUS driver, and is supposed to be unit-specific. Value values are:

| | RegMode | Order of bytes |
|---|---|---|
| Big endian (default) | 0 | 1 0 |
| Little endian | 1 | 0 1 |

Any other value reverts to big endian.

Parameter type: integer

Default value: 0

Examples:

[PWRMODBUS]

RrMode = 0 # Default

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

RegMode = 1 # This device has little endian registers


© 2011 Schneider Electric All Rights Reserved

## timeZone

Time zone names are taken directly from the Windows registry database (case-insensitive), and will otherwise default to using the I/O server's local time zone. The Windows time zone database is located in the Windows registry in:

HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\Time Zones\. The examples of time zone names are:

AUS Central Standard Time

China Standard Time

Pacific Standard Time (Mexico)

Use the general section [POWERLOGICCORE] to specify the time zone for all devices, e.g.

[POWERLOGICCORE]

Timezone = Mountain Standard Time

This sets the default time zone for all devices (Sepam, PLogic, Micrologic, PWRMODBUS). Otherwise the time zone can be specified for each device with precedence taken as described in the start of this section.

Examples:

[PLOGIC870.Cluster1.Singapore_Port]

Timezone = Singapore Standard Time

[PLOGIC870.Aus_Cluster]

Timezone = Aus Central Standard Time

Not having a time zone specification means that the device is in the same time zone as the machine where the I/O Server is running. No time conversion will be done.

© 2011 Schneider Electric All Rights Reserved

# Performance Tuning Parameters

Several parameters are provided to allow tuning of the performance. These parameters fall into three broad categories; bandwidth allocation, packet blocking optimisation, and tag scan rates.

## Bandwidth Allocation Parameters

Bandwidth can be allocated for the different types of data as desired. The parameters to perform this are as follows:

| [Parameter] | [Default Value] | [Parameter Type] |
|---|---|---|
| EventBandwidth | 25 | integer |
| WaveformsBandwidth | 12 | integer |
| CommandsBandwidth | 13 | integer |
| RealTimeBandwidth | 50 | integer |

The percentage bandwidth allocated to each queue will be the ratio of an individual queue's value when compared to the total sum of defined bandwidths. The default values have a sum of 100 for ease of reference. Any unused bandwidth will be shared amongst the other categories.

Bandwidth can be configured at the port level, but not the device level.

Example:

[SEPAM40]

EventsBandwidth 30

WaveformsBandwidth 5

CommandsBandwidth 15

RealTimeBandwidth 50

[SEPAM40.MYCLUSTER.PORT_1]

EventsBandwidth 50

WaveformsBandwidth 30

CommandsBandwidth 10

RealTimeBandwidth 10

**BandwidthAllocation**

This parameter allows the ratio of bandwidth assigned to each device sharing a port to be configured. This parameter can only be configured at the device level.

Parameter type: integer

Default value: <equal split>

Example:

[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]

BandwidthAllocation 70

[SEPAM40.MYCLUSTER.PORT_1.DEVICE_B]

BandwidthAllocation 30


© 2011 Schneider Electric All Rights Reserved

## Packet Blocking Optimisation Parameters

For all devices except the Sepam, parameters can be configured to optimise the MODBUS packets that are created for collection of data from the device. Sepam devices have pre-configured blocks that are already optimised.

The parameters that control the blocking are as follows:

**enableScatteredReads**

This causes the driver to use the 'scattered read' extension that can help improve blocking. This option should be enabled for devices that support this extension.

Parameter type: Boolean flag

Default value: 0 for generic Power MODBUS driver, 1 for PowerLogic driver

Example:

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

enableScatteredReads 1

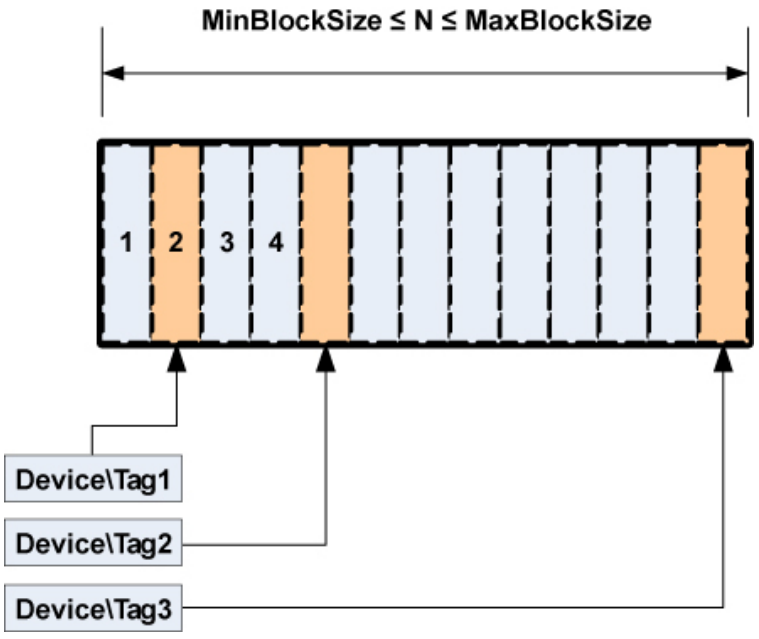
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_B]

enableScatteredReads 0

**percentBlockFill**

This parameter defines the maximum percentage of configured registers contained in a block before the drivers creates fixed blocks instead of scattered blocks. The following figure illustrates now a block of N registers can be constructed:

- If M<N registers are configured, the block builder can either create a scattered block or a multi-register block.
- If M/N*100% is less than PercentBlockFill, the block builder creates a scattered registers block.
- If the percentage of configured registers ≥ PercentBlockFill, the block builder creates a multi-register block.




Parameter type: percentage

© 2011 Schneider Electric All Rights Reserved

Default value: 50

Example:

[PM870.MYCLUSTER.PORT_1.PM_DEVICE]

percentBlockFill 50

[CM4000.MYCLUSTER.PORT_1.CM_DEVICE]

percentBlockFill 80

**maxBlockSize**

This parameter defines the maximum number of registers that can be read in a single request. By default, this is 124, but some devices can read more than this.

Parameter type: integer

Default value: 124

Example:

[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]

maxBlockSize 1024

**minBlockSize**

This parameter defines the minimum number of registers to read as a fixed block before the block builder will instead add those registers to a scattered block. If latency is low, and scattered reads are expensive, this value should be lower. If latency is high, or scattered reads are inexpensive, it is better to set this value higher. Only applicable when scattered reads are enabled.

Parameter type: integer

Default value: 20

Example:

[PM870.MYCLUSTER.PORT_1.LOW_LATENCY_DEVICE]

minBlockSize 10

[CM4000.MYCLUSTER.PORT_1.HIGH_LATENCY_DEVICE]

minBlockSize 100

### Tag Scan Rate Parameters

Each tag can be configured at a priority level from 1-3 where 1 is the highest. Parameters exist to adjust the relative scan rates of the high and low priority tags in comparison to the nominal tag scan rate.

**HighScanRate**

Parameter type: percent relative to nominal

Default value: 50

**LowScanRate**

Parameter type: percent relative to nominal

Default value: 200

Using the default parameters, the high priority tags will be refreshed twice as fast as the normal priority tags, and the low priority tags will be refreshed at half the rate of the normal priority tags. These parameters can be configured at the port level and higher.

Using the default settings and a nominal tag refresh rate of 1 second:

Low Priority Tag Refresh: 2000 ms


© 2011 Schneider Electric All Rights Reserved

Normal Priority Tag Refresh: 1000 ms

High Priority Tag Refresh: 500 ms

Example:

[PM870.MYCLUSTER.PORT_1]]

HighScanRate 25

LowScanRate 500T

## Advanced Tag Block Capabilities (Invalid Memory Access Blocks defined)

Some devices may restrict access to certain memory registers. Such registers may be available for read only, write only or may not be available at all, resulting in a MODBUS exception when the registers are addressed.

Definition: Blocks of registers that cannot be read or written to are referred as “invalid memory access blocks.”

These devices create a challenge for the PWRMODBUS driver. If the device has invalid blocks that do not support scattered reads (or they are disabled for this device), the driver may try to read registers in blocks that intersect with the registers that cannot be read. This can result in the whole block being invalidated and, in certain cases, may also result in device being taken offline. Figure 1 (below) illustrates an invalid block in the middle of an address space.

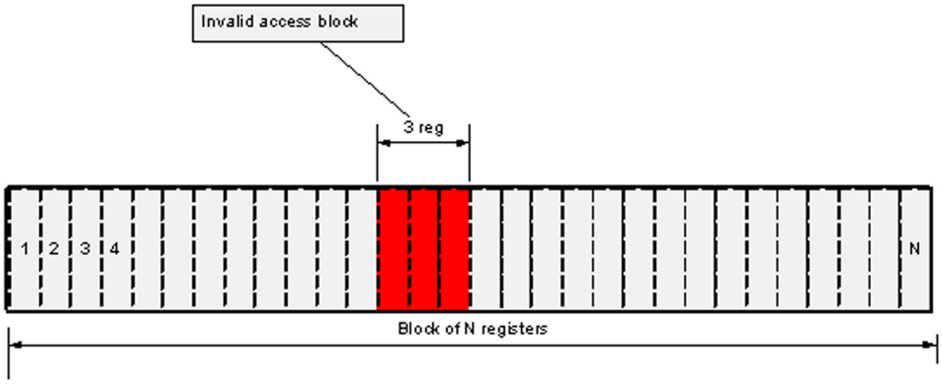
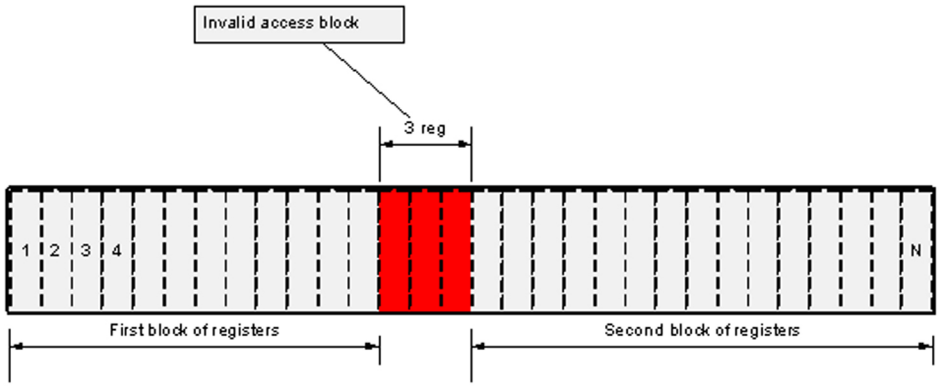



Figure 1 shows how the "invalid memory access block" affects MODBUS register blocking. In this situation, if the driver does not know that the block of 3 inaccessible registers exists, it will try to block all registers from 1 to N (depending on data that was requested by the real-time data collector). This block, however, will never be read successfully, as the device will respond with an exception to all attempts to read invalid registers.

If the configuration includes information about invalid memory access blocks, the driver will create two blocks instead of one, as shown in Figure 2:

© 2011 Schneider Electric All Rights Reserved




In Figure 2, invalid registers were taken into account when the block was constructed. When configuring device that has invalid memory areas, it is especially important to define all blocks that may interfere with any of the tags.

### Invalid Block Tag Definition Syntax

Invalid access memory areas are defined as variable tags, using the following format for the address:

```
T:IB;{m|i|c|s}:<start_register>:u<count>;E:1;L:P:0
```

where

- `m, i, c` and `s` define the type of MODBUS register
- `<start_register>` is the first register address of the invalid access block
- `<count>` defines the number of registers in the invalid access block

Example:

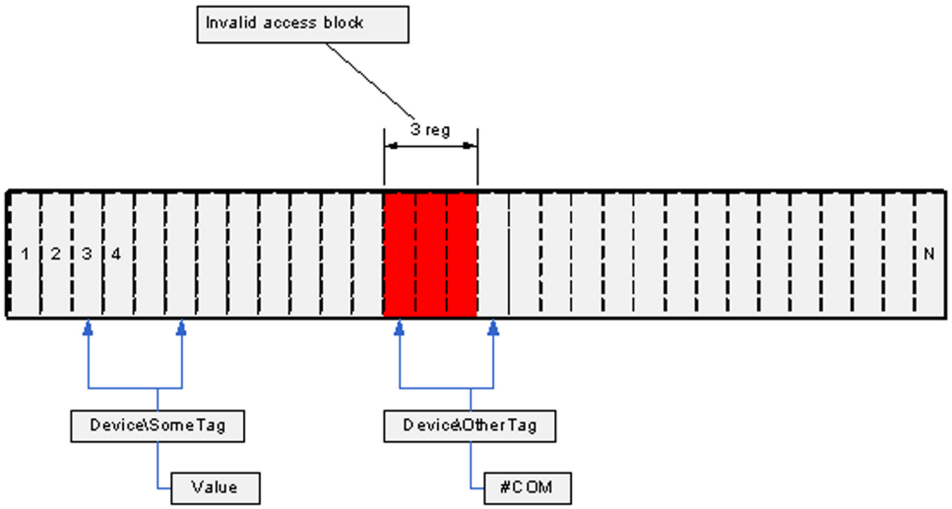
```
T:IB;m:300:u10;E:1;L:P:0
```

This defines an invalid access block of ten holding registers starting from register 300.

### Configuration Notes

When one or more invalid access blocks is defined according to the syntax above, tags configured to read any of invalid registers will be affected by it. If any of the tag registers fall into an invalid memory access block, this tag will not be readable; any attempt to read its value will result in #COM, as shown on Figure 3:


© 2011 Schneider Electric All Rights Reserved




However, such tags do not affect other tags, because the PWRMODBUS driver implements algorithms that prevent tags from being invalidated by invalid memory block logic.

Tags that try to use invalid registers are detected on startup and can be found by analysing the log file. This is an example trace:

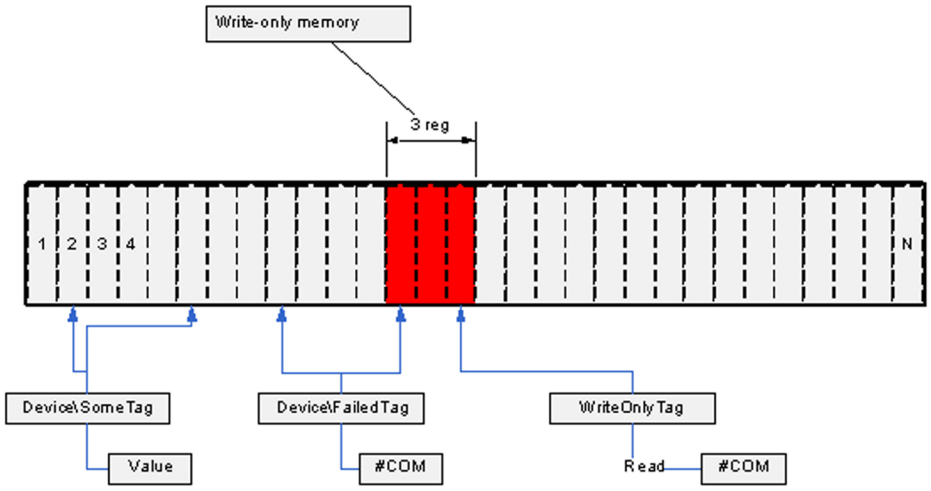
```
[DEBUG]
[REAL][GeneralDriver::BaseDatapointBuilder::BuildDataPoints()]
Adding datapoint. Tag - BCM1\H_QIVR34\SwlStr Address -
T:SS;m:283:2;E:1;L:P:26 Datapoint: class Datapoints::Status_SS

[DEBUG] [REAL][RealTimeData::DeviceCache::Subscribe()] Init
Registers: Polled Registers: Address:283 Type:3

[ERROR] [MISC] [RealTimeData::BlockBuilder::AddDataPoint()] Cannot
add datapoint, one or more invalid memory addresses fall into non-
splittable block

[ERROR] [MISC]
[GeneralDriver::BaseDatapointBuilder::BuildDataPoints()] Could not
init datapoint. Tag BCM1\H_QIVR34\SwlStr Address
T:SS;m:283:2;E:1;L:P:26. Analyze other messages, this tag address
may contain invalid registers
```

Such output is expected when a holding register with address 283 is declared invalid. This trace helps figure out any configuration issues.

### Write-only Memory

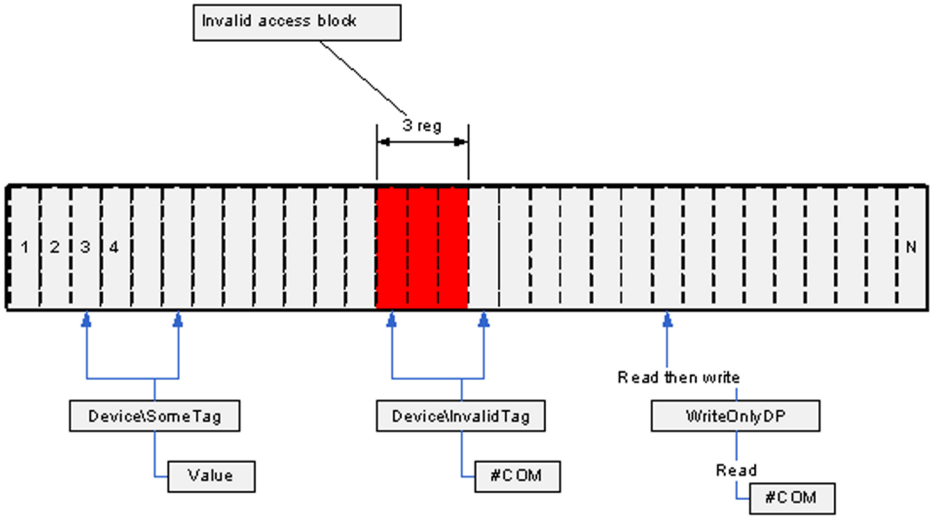
If a certain register range is accessible for write only, no additional configuration may be needed. However, to prevent the driver block optimizer from including these registers in a poll, they also must be configured by adding “invalid block” tags as described in the *Invalid Block Tag Definition Syntax*, described above. Declaring these registers invalid will not prevent drivers from trying to write to them. Figure 4 illustrates a write-only memory configuration:

© 2011 Schneider Electric All Rights Reserved




Write-only registers should not be confused with write-only datapoints that internally read a register before attempting to write. Declaring the register they read invalid will result in a datapoint not working; such mistake should be avoided. Figure 4 shows “WriteOnlyDP” as an example; this tag cannot be read (it will result in #COM), but internally it needs to get the register value before writing into it. If this register was declared invalid, tag writes would also fail.

Figure 5 illustrates a write-only datapoint:




## Tag Blocking Notes

PowerLogic SCADA drivers support an advanced blocking mechanism for tags. That is, real-time tags are no longer blocked together with write-only tags.

Tags found invalid, due to intersection with invalid memory areas, are not blocked with “good” real-time tags and will not therefore interfere with them.

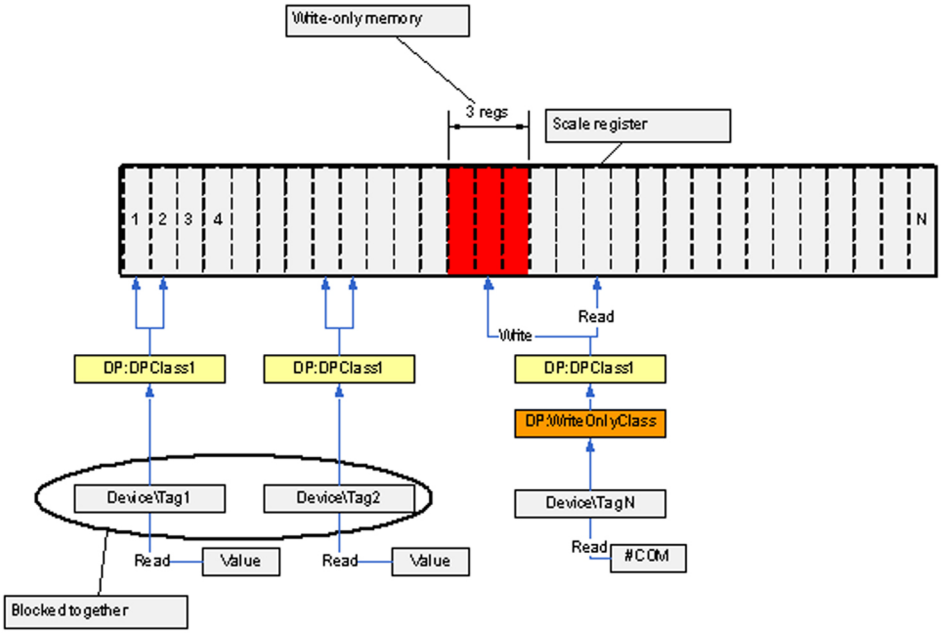
## Write-only Tags

Beginning with driver version 2.0.1.1, the write-only tags feature is fully supported.


© 2011 Schneider Electric All Rights Reserved

There are no special logic codes or address formats for write-only tags. If a tag references memory that was declared invalid (see *Invalid block tag definition syntax*, above), and its datapoint has writing capabilities, the tag becomes write only. No preliminary checks are performed to verify that the memory can be written to, and no additional configuration is needed. It is assumed that, if the tag is configured to write into memory that has been declared “invalid,” the memory can actually be written to.

It is important to understand that scaled write tags (code 110) will become write-only tags, if that scale register can be read. Tag Device\TagN on Figure 6 explains this case: the datapoint needs to read the scale value from the scale register in order to write scaled value to write-only register. However, as long as the actual register belongs to the memory that can only be written to (and it is configured using T:IB tag syntax as explained in *Invalid Block Tag Definition Syntax* above), this tag cannot be read.




The fact that the tag mentioned before cannot be read will not affect other tags reads (see *Tag Blocking Notes*, above).

Note: The write-through feature of the device cache is disabled for write-only tags.

© 2011 Schneider Electric All Rights Reserved

# Waveform Parameters

The following parameters configure the waveform downloading behaviour. These parameters are only applicable for Sepam devices and PowerLogic devices that support waveforms.

[Parameter] [Default Value] [Parameter Type]

WaveformsDisable 0 Boolean value

WaveformMatchMargin 10 seconds

WaveformCheckTime 30 seconds (PM/CM)

WaveformZone 1 integer (Sepam)

## WaveformsDisable

This parameter enables or disables waveform downloading for a particular device.

Parameter type: Boolean value

Default value: 0

Example:

[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]

WaveformsDisable 1 //Disable waveform downloading

NOTE: This INI setting is a global setting that sets the default at startup. You can set this for any set of devices (clusters, individual devices, etc.)

There is also a tag that will change an individual device's setting at runtime (it will reset to the default when you restart the project). This tag is LLNO\WaveformCollectionEnabled. 1 = True, 0 = False.

## WaveformMatchMargin

Alarms are matched to waveforms by the timestamp of each. This parameter is the maximum difference between alarm timestamp and waveform timestamp for PowerLogic SCADA to consider it a match.

Parameter type: seconds

Default value: 10

Example:

[SEPAM40]

WaveformMatchMargin 2

## WaveformCheckTime (PM, CM, and Sepam)

This parameter defines the time the driver will wait between checking for new waveforms.

Parameter type: seconds

Default value: 30

Example:

[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]

WaveformCheckTime 60 (checks every 60 seconds)


© 2011 Schneider Electric All Rights Reserved

## WaveformZone (Sepam)

This parameter defines the Sepam waveform zone that the Sepam driver will use to collect waveforms from the device. This allows two masters to extract waveforms from the same device. Valid values are 1 or 2.

Parameter type: integer

Default value: 1

Example:

[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]

WaveformZone 2

© 2011 Schneider Electric All Rights Reserved

# Sepam Event Reading Parameters

These two Sepam parameters are described below.

## EventTable

This parameter defines the Sepam event table that the Sepam driver uses to collect alarms from a device. This allows two masters to extract alarms from the same device. Valid values are 1 or 2.

Parameter type: integer

Default value: 1

Example:

[SEPAM40.MYCLUSTER.PORTO_1.DEVICE_A]

EventTable 2

## EventIdle

This parameter defines the time that the driver will wait before requesting the next event from a Sepam device. It may be possible to reduce this value to increase the rate at which alarms can be retrieved from the device.

Parameter type: milliseconds

Default value: 500

Example:

[SEPAM40.MYCLUSTER.PORTO_1.DEVICE_A]

EventIdle 200


© 2011 Schneider Electric All Rights Reserved

# MicroLogic Modules Configuration Parameters

A MicroLogic unit consists of three or four modules, each acting as a separate MODBUS device; however the I/O server views MicroLogic as one I/O device. The communication control module (CCM) is optional for MicroLogic; its presence may be detected by the driver or specified in the INI file.

## CCM

The CCM parameter specifies whether a CCM is present on the device or if the driver should try to detect its presence ("auto mode"). Valid values are:

CCM not present - 0

CCM present - 1

Auto mode - 2 (default)

Any other value reverts to auto mode.

Parameter type: integer

Default value: 2

Example:

[Micrologic.MYCLUSTER.PORT_1.DEVICE_A]

CCM=1

## Module-Specific Packet Blocking Optimisation Settings

Due to different firmware versions, MicroLogic modules may require different blocking settings. This is especially true when MicroLogic contains a BCM that supports MODBUS "read multiple registers" requests for up to 124 registers, and an MM or a PM module that supports 21 register reads at max. The MicroLogic driver allows blocking optimisation parameters to be overridden for each of the device's modules, as in the following example:

[Micrologic.MYCLUSTER.PORT_1.DEVICE_A]

maxBlockSize = 124

[Micrologic.MYCLUSTER.PORT_1.DEVICE_A.BCM]

maxBlockSize = 21

The parameter set for the device applies to all of its modules unless overridden in a module-specific section (e.g., [Micrologic.MYCLUSTER.PORT_1.DEVICE_A.BCM])

These parameters can be overridden:

- enableScatteredReads
- minBlockSize
- maxBlockSize
- PercentBlockFill

This applies to the BCM, CCM, MM, and PM modules.

© 2011 Schneider Electric All Rights Reserved

# Data Replication Parameters

These parameters are used to configure the data directory paths of your servers. These settings are server wide, and must be added to the 'WaveformDB' area of the ini file.

## Database root folder path

Waveform databases for all units will locate on the filesystem under the same common folder. The path to the root folder will be specified in the citect.ini file:

[WaveformDB]
LocalRoot = c:\path\to\the\database\root

This path must be specified as local path.

By default, PowerLogic SCADA [DATA]: directory is be used as database root folder.

## Database root UNC path

For waveform files to be accessible by the remote clients, the database root folder must be available as network shared folder. The UNC name of this folder must be specified in the INI file

[WaveformDB]
UNCPath = \\computerName\shareToTheLocalRootAbove

If the UNC path to the database root is not specified, all waveform file names returned by the library will be local file names for the I/O server, making viewing the waveforms on the remote clients impossible.

## Replication destination configuration

In redundant scenario, the replication target folder must be specified for replication to work

[WaveformDB]
ReplicationDestinationRoot=\\OtherMachine\share\path

The destination path is the name of the network share on the redundant machine where its waveform database root is located. It must also allow write access.

No default value for it is assumed.

If not set or share is not accessible, no replication will be preformed.


© 2011 Schneider Electric All Rights Reserved

# Graphics Library Parameters

## Maximum number of entries that can be held in Event Log

The Alarm Summary length parameter in Citect.ini defines the maximum number of entries that can be held in the Event Log (default = 5000 entries). You can view all events in the Event Log and alarms in the alarm logs (Alarm Log, Unacknowledged Alarms, Disabled Alarms).

Each event requires 256 bytes of memory, plus the length of the comment. 32,000 entries will require at least 8 MB of memory. If you have many events, you should ensure that there is enough memory to store them in RAM.

After the parameter number is reached, older events are FIFO'd out to storage in [Installed Project Directory]\Schneider Electric\PowerLogic SCADA 7.20\Logs

## Parameters for Alarm and Event States

**[Alarm]**

UseConfigLimits = 1

CacheLength = 2500

!Sound1 = <wave file name>

!Sound1Interval = <repeating interval in milliseconds>

!Sound2 = <wave file name>

!Sound2Interval = <repeating interval in milliseconds>

!Sound3 = <wave file name>

!Sound3Interval = <repeating interval in milliseconds>

**[AlarmFormat]**

EventLog=OnDate | Date, OnTimeMS | Time, Custom1 | Equipment, Name | Description,
SumState | State | Custom2 | Location, UserName | User

**[AlarmStateText]**

ON=<default text for ACTIVE state>

OFF=<default text for INACTIVE state>

ACK=<default text for ACKNOWLEDGED state>

ENA=<default text for ENABLE state>

DIS=<default text for DISABLE state>

CLE=<default text for CLEAR state>

These parameters are read only when the system starts up. The user must restart PowerLogic SCADA if they change these parameters.

If you do not specify any value for these parameters, these default values will be used, in this order: .

Appearance

Disappearance

Acknowledge

Enable

Disable

Clear

**[PLSEventLog]**

© 2011 Schneider Electric All Rights Reserved

FileFormat=0

EntryCount=250

ScanRate=10

[Path]

[PLSEventLog]=c:\SharedData\Alarm

**[General] IODevCheckStartupDelay**

Delay time before the I/O server starts checking for I/O device status at start-up. The delay allows time for the I/O devices to come online. Otherwise, the I/O server would have triggered communication failure alarms for the relevant equipment.

Allowed Values: ≥ 0

Default Value: 0

**[General] IODevCheckInterval**

The time interval in seconds that the I/O server repeats the I/O device status check.

Allowed Values: ≥ 2

Default Value: 2


© 2011 Schneider Electric All Rights Reserved

# Appendix 3: Additional INI Parameters

The following parameters are added to support device types as they are added to the system.

### Quantum PLC Time-Stamped Events

The PWRMODBUS driver supports Quantum time-stamped events. You must set the following INI parameter to enable time-stamped alarms downloading:

[PWRMODBUS]

TSEventsEnabled = 1
0 by default, valid values 1 or 0

TSMailboxAddress = 1104
1104 by default

TSAddrLost = 705
705 by default

© 2011 Schneider Electric All Rights Reserved

# Appendix 4: Glossary

### address

The address contains all the information the SCADA system needs to get values from a known device, and to return a value determined by the values read from the device and the calculation rules defined in the address.

### alarm categorization

Added when setting up custom tags, this is one of the alarm filters. which will be used for filtering and sorting alarms in the Alarm Log. Categories are: normal, over, over hs, rate of change, reversal, sag, swell, transient, under, and under hs.

### alarm text (On/Off)

For onboard alarms, this is the text (added while adding a custom tag) that displays when the alarm is on or off. This text will display in the Alarm Log.

### alarm filters

Setup in the Profile Editor, these filters help you filter and sort data that displays in the Alarm Log.

### alarm groups

Added when setting up custom tags, this is one of the alarm filters. which will be used for filtering and sorting alarms. Groups are: frequencies, motors, power factors, powers, temperatures, time, and voltages.

### alarm levels

Added when setting up custom tags, this is one of the alarm filters. which will be used for filtering and sorting alarms. Levels are: event, high, medium, and low.

### alarm types

Added when setting up custom tags, this is one of the alarm filters. which will be used for filtering and sorting alarms. Types are: diagnostic, power quality, protection, and system.

### Automation Interface

Used instead of the Profile Wizard, this tool allows you to add multiple devices at a time to a PowerLogic SCADA project.

### bandwidth

The amount of space or processor resource being used by a part of the system. You can use the bandwidth allocation parameters to allocate bandwidth for different types of data.

### baud rate

The speed of transmission of electrical signals on a line. This is often described in bits per second (bps), although the baud rate and bps are not truly interchangeable. The


© 2011 Schneider Electric All Rights Reserved

baud is actually the measurement of how frequently the sound changes on the line.

### bitmask

A mask is defined as data that is used with an operation to extract information that is stored in another location of the code. A bitmask is the most common mask used. It extracts the status of certain bits in a binary string or number (a bit field or bit array).

### Cicode

This programming language, which is similar to Visual Basic or "C," allows you to access and edit real-time data in the PowerLogic SCADA project. Although not difficult to use, the person working in Cicode must have received Cicode training from PowerLogic SCADA.

### cluster

A discrete group of alarms servers, trends servers, reports servers, and I/O servers. It would usually also possess local control clients. For a plant comprising several individual sections or systems, multiple clusters can be used, one cluster for each section.

### Color Change Tool

(also Project Color Change Tool) This tool allows you to assign colours for busbars and circuit breakers in genies, for use in graphics pages. You can assign separate colours for up to seven different states (for example, error, intermediate). When the busbar or circuit breaker is in the specified state, the system will change its colour to match what is set up here.

### CommsMethod (communications method)

This is the communication protocol, such as MODBUS/RTU via Gateway, that is being used by a device. When adding devices in the automation interface, you will need to specify the CommsMethod.

### ComPort

(also COM port) The computer's communications port used to connect to devices, for sending and receiving serial data.

### composite device type

A composite profile can be made from more than one device type. Each device type included in the composite profile can use its own protocol for communication. The composite device type allows the engineer to use two devices for one monitoring point, e.g., a breaker and a monitoring device. PLSCADA combines the functionality of the two devices so that the end user only needs to consider one device when analysing that location in their system.

### configuration environment

(See design time environment.)

### control

This is a command written to a device register that then causes an action within some equipment. There are a series of default control tags in PowerLogic SCADA to achieve

© 2011 Schneider Electric All Rights Reserved 

these actions. For example, in the Sepam 40, there are control tags to operate a circuit breaker and enable a recloser.

### custom device type

This is a "new" device type that is added to a system. Although the Profile Editor includes many standard device types, it may be necessary to add a new device type that includes custom tags, or one that includes a different set of tags than the standard device types.

### custom tag

This is a "new" tag that is added to the system. Although the Profile Editor includes many standard tags, you may need to add a tag for a third party device, or to edit an existing tag to change its attributes. In these cases, you need to add a custom tag. These tags are then added to a customized device type to be made available in profiles and projects. The custom tag creation interface applies rules to the tag creation to help guide the user to making tags that will correctly retrieve the desired information from devices.

### DataBits

This is the number of data bits used in data transmission. The I/O device and the ComPort must have the same value.

### data type

Data types are restricted to these types that are supported by the SCADA system: digital, int, long, real, and string.

### demo mode

This demonstration mode allows you to run PowerLogic SCADA without a hardware key. You can use all of the PowerLogic SCADA features, but with limited runtime and I/O options. See *Using the Demo Mode* on page 157 for details.

### design time environment

To be used only by the person who is creating and installing the project for the end user, this is the environment in which you add devices, profiles, and projects, as well as create genies and one-lines.

### device category

Used in the Profile Wizard to logically group device profiles, to make them easier to locate. The default category is "Schneider Electric, and the default subcategories are "Monitoring Device," "PLC," and "Protective Device." Do not confuse these terms with:

- categorization and subcategorization (alarm filters, used during runtime, to filter and sort alarm data)
- category type: real-time filters that provide metadata for future reporting

### device profile

A subset of the device type: where the device type includes all of a device type's attributes, the device profile includes only the specific tags that are used by an individual customer. A device profile is set up like a device type, except that it is specially configured for a particular need. For example, a CM4000 that is being used to monitor the main at a given facility would have a different profile from the CM4000 that


© 2011 Schneider Electric All Rights Reserved

is used to monitor water and gas at a facility. The profile also allows you to designate that some tags will be used for trending and/or for PC based alarming.

## device type

A device type contains all the information for retrieving the available information from a given device type. This information is stored in the form of tags. Tags can be of these types: real-time, onboard alarms, controls, and Resets. Real Time tags can be further separated into groups such as Currents or Energies.

A device type has a name and has one or more drivers associated with it. It also has one or more tags associated with it; for each driver/tag combination, the device type can have an address.

## device type drivers

These are programs that allow PowerLogic SCADA to interact with a device or series of devices. PowerLogic SCADA includes several generic drivers (generic MODBUS, Sepam 40 Range, MicroLogic 5P and 6P, CM4000 series, and PM800 series) that interact with "standard" device types.

## engineering unit templates

Used for conversions between base units and their conversions (for example, inches to centimeters or amperes to kiloamps).

## enumeration (used for the circuit breaker status)

This is a single value (0-5) that defines a condition that is determined by multiple bits. They allow for dynamic contingencies, such as when you need to use multiple bits to describe the position of a circuit breaker.

## format code

These codes define the attributes of the address field of a tag. See *Format Code Definitions* on page 80 for a list of format codes.

## functional addressing

Creates addressing for a device that has data residing in different registers. Functional addressing dynamically addresses the device, based on its configuration (using C#, you can write code to account for user-defined variables). When you add the profile to a project, you will enable functional addressing. Then, when exporting to the Profile Wizard, you are prompted for the variable(s) related to these device types.

## genie

A genie is a multi-layer graphic that is used on the Graphics page to indicate an object, such as a motor, generator, circuit breaker, or switch. Using genies, you only have to configure common behaviours of that object once. The default genie library includes a large number of pre-defined genies. A graphics page can contain any number of genies.

## graphics test mode

Only available in the pls_example project, use this mode for off-line testing of animation in one-line drawings. Do not use this mode with physical devices that are connected to a live environment.

© 2011 Schneider Electric All Rights Reserved

## ICD file

*IED capability description*: This is the file that is imported iinto the Profile Editor from an IEC 61850 device. Editing for ICD files is limited to the ability to add/delete datasets and control blocks, and the ability to edit buffered and unbuffered control blocks that were created in the Profile Editor.

## IEC tag name

The IEC 61850-compatible name that is created when a tag is created. This is the name that is used by the SCADA system. The tag names provided use an abbreviated form of the IEC 61850 naming convention. A tag name cannot include any special characters except ( _ \ ). It can be a maximum of 32 characters.

## IED

*Intelligent electronic device*

## IID

*Instantiated IED description*: defines the configuration of one IED for a project; is used as the data exchange format. This file contains data for just the IED that is being configured.

## logic code

Logic codes tell the PowerLogic SCADA program how to mathematically certain values in device registers, thus providing values that the user needs. Examples of logic codes are date and time for a circuit monitor or a Sepam device, digital inputs/outputs, and IEEE power factor.

## metadata

Metadata provides data about other data. In PowerLogic SCADA, metadata might include additional information about a custom tag: its category type, utility type, statistical type, or quantity. It is often used for reporting purposes.

## multi-monitor support

This option allows you to view the runtime environment from multiple computer monitors. In PowerLogic SCADA, this allows you to view a different startup page on each monitor.

## onboard alarm

Onboard alarms are alarms that are detected and stored in a device's data logs. If an onboard alarm is configured within a device, you can map it, via the Profile Editor, to a digital time-stamped alarm in PowerLogic SCADA. These alarms and associated waveforms can be read and displayed in the PowerLogic SCADA Alarm Log.

## parity

Parity is used as a simple means of detecting error by verifying that the result is odd or even. In PowerLogic SCADA, parity is required for the generic serial or MODBUS/RTU comms methods, when adding a device to PowerLogic SCADA.


© 2011 Schneider Electric All Rights Reserved

## PC-based alarms

PC-based alarms are alarms that are detected from a device and are stored in the software. You can add them to the Profile Editor when you create the device profile. All PC-based alarms are analog by default.

## PMCU

The Meter Configuration Help Utility. Use this application to set up the features within PowerLogic devices, and enabling such features as onboard alarms and waveforms. The information that is generated from PMCU is then available for use within PowerLogic SCADA.

## point (see SCADA tag)

## polling priority

When adding a custom tag, this field determines the level of priority that PowerLogic SCADA uses when reading data from the related device. Options are low, normal, or high.

## PowerLogic SCADA tag name library

This library includes electrical parameters, or measurements or topics. A tag name has three parts:

- an easy to read name (such as Current Phase A)
- a unique identifier
- meta data (attributes used to categorize the data for intelligent display/analysis)

## Profile Editor

This tool allows you to create device type tags, device types, and device profiles. This information is then imported into PowerLogic SCADA, for use in creating graphics pages.

## Profile Wizard

This tool allows you add device profiles to, or delete them from, a PowerLogic SCADA project. From the Profile Editor, you export profile data into a file that can be used in PowerLogic SCADA. From there, you use the Profile Wizard to add the device profile into a PowerLogic SCADA project.

## project

A project is made up of any number of profiles. Profiles that have been added to a project can be imported into the SCADA system and made available for setting up actual devices in the SCADA system.

A project name must match exactly between the Profile Editor and Citect SCADA.

Each project includes: a unit template, display name, and one or more instantiated device profiles (instantiated by choosing a device profile and specifying a name). The following is a simple example of how device profiles and projects inherit information from the device type.

- The device type **myCM4Type** can use either the Modbus driver or the IEC 61850 driver.
- The device profile **myCM4Profile** inherits this device type.

© 2011 Schneider Electric All Rights Reserved

- The project **myCM4Project** instantiates the **myCM4Profile** and calls it **myModbgusCM4**, and it specifies that it uses the Modbus driver.
- When this project is imported into the SCADA system, Modbus addressing will be used.

## Project Color Change Tool

This tool, accessed from Citect Explorer, allows you to set unique colours for seven different states in busbars and circuit breakers.

## register scaling

This is a conversion that is the result of applying a scaling multiplier to a register value.

## resets

This feature allows you to reset data from a device. There are some pre-defined resets, such as device date/time and onboard data logs, You can also add custom resets.

## reserved names

The following terms are reserved for use in the Include project. If you use them in projects that you create, they can cause compilation errors:

- IO_Server
- Report_Server
- Alarm_Server
- Trend_Server
- Client

## runtime environment

This is where the end user views system information. This environment includes the one-line diagrams with interactive objects, alarm and event pages, and analysis pages (from which users can view trends and waveforms).

## SCADA tag (SCADA point)

A SCADA tag is an extension of the tag name. A SCADA tag is made up of five parts: two in addition to those already defined in the PowerScada tag library:

- an easy to read name (such as Current Phase A)
- a unique identifier
- an address (where on a device to read the raw data from)
- a formatting scheme (what to do with the data after it is read to scale it)
- meta data (attributes used to categorize the data for intelligent display/analysis).

## SCL

*Substation Configuration Language*, the configuration description language for communication in electrical substations related to IEDs (defined by IEC 61850-6). This language is used when importing/exporting ICD files. SCL files are used in such devices as G3200 gateways.

## StopBits

The number of bits that signals the end of a character in asynchronous transmission. The number is usually 1 or 2. Stop bits are required in asynchronous transmissions because the irregular time gaps between transmitted characters make it impossible for the server or I/O device to determine when the next character should arrive.


© 2011 Schneider Electric All Rights Reserved

## super-genie

Dynamic pages (usually pop-ups) to which the system can pass information when the runtime page displays. You can use super-genies for pop-up type controllers (for a very specific task that may not be always needed).

## tag

Any quantity or measurement (topic) that is recorded by the device; for example, current A. All tag names will use the IEC61850 naming convention. The user can create custom tags; the naming convention will be in the following format:

<EquipmentName>\<PointName>

Where <EquipmentName> uses '_' (underscore character as a separator)

Where <PointName> uses '\' (backslash as a separator)

For example: SST_MV_BUSA_INC1\XCBR1\Pos

A tag contains a tag description, units, tag name, data type, and address.

Tags include the following (* indicates required information):

tag name*
display name*
group*
data type*
engineering units
Citect formatting
polling priority
alarm "on" text
alarm "off" text
category type
utility type
statistical type
quantity
alarming categorization
alarm type
alarm group
alarm subcategorization
alarm level

The tag's group determines the tag's class:

If the tag's group is onboard alarm, control, or reset, the tag's class is the same.

If the tag's group is anything else, the tag's class is real time.

## tag address

This "address" includes everything you need to know about a tag (quantity/topic). Included are the data type, priority, and logic code; and how the tag is displayed in registers. You can change address attributes on the Edit Address screen. The full tag address displays on the Define Device Type Tags tab when "Advanced Properties" is selected.

© 2011 Schneider Electric All Rights Reserved

## tag description

The tag description is a human readable name which can include spaces and special characters (except for \ / : * ? < > | ). The description can be a maximum of 32 characters long.

## tag group

The basic groups include: real-time, of which there are several sub-groups (for example, currents, energies, frequencies and power factors); onboard; control; and reset.

## units

Units are the standard measurement associated with the quantity measured by a tag. Units come in two types: base units and conversion units.

Some information is common to all all units, and some applies only to conversion units:

**Common Information**: base unit name, base unit abbreviation

**Conversion Unit Information**: conversion unit name, conversion unit abbreviation, offset, multiplier

## units template

The units template defines the conversion factor that must be applied to the standard units provided in order to give the user their information in their desired units. The units profile applies to an entire project. For example, iIf the standard unit for a device is MW, but the user wants their project to display KW, they must define this units conversion in the units template and then apply it to an entire project.

## user privileges (user access, user rights)

This feature allows you to control the amount of access that each user has to the system. User privileges are password-protected. See*Default User Access Settings (Privileges)* on page 159 for more information.

## vector math

Vector math and vector math IEEE are two logic codes. They are the result of math that use vectors, which are directed quantities.

## voltage levels

Voltage levels are used in the Color Change Tool. There are four levels (0–3). The measurement of each of these levels corresponds to a level used by the standard that applies for each customer.


© 2011 Schneider Electric All Rights Reserved

# Appendix 5: Frequently Asked Questions (FAQs)

The following items provide information about topics that generate frequent questions.

## How do multiple engineers create device types and profiles at the same time?

It is highly recommend that one PC be nominated as the Master Profile Editor machine. Until the import/export has been incorporated into the Profile Editor, customisation must be completed at one PC. Then you can transfer the data to other machines. Not every change needs to be made on this machine – only changes that will impact your ability to share device types and profiles.

You must ensure that you have maintained synchronization between all the computers for:

- units and unit templates
- categories and sub-categories
- custom tag names

Recommendations on how to do this include the following:

1. Nominate one machine as the master.
2. Have a method in place to create the customisation on the master machine.
3. Nominate one person to make the changes, or allow multiple people to access the master machine and make their own changes.
    a. Ensure that everyone who needs access to the machine has access. It would be ideal to have something similar to Remote Desktop available to your team. Whatever method you use, ensure that only one person can be logged in at a time.
    b. Users must log in to the master machine, make their changes/custom entries, and then synchronize their local machine.
4. Have a daily backup in place for the master machine. Save several weeks of backups before overwriting them.

## If I don't use PowerLogic drivers, how do I create device profiles?

The easiest way to do this is to:

1. Create a device type using a non-PowerLogic driver (like MODNET).
2. Using that device type, create a device profile.
3. You need to change the addressing of the new device type. The easiest way to do this is to copy the addressing from a known device type, and then make the necessary changes for the new device type.

## Can I install ION E or SMS on the same computer as PowerLogic SCADA?

We do not recommend this practice. Each of those applications uses SQL server, which requires so much of the CPU that PowerLogic SCADA performance would be severely degraded.

© 2011 Schneider Electric All Rights Reserved

## When creating custom tags, how do we ensure consistency in their names and attributes?

Until the Profile Editor has been expanded to include an import/export utility, it will take work on part of the integration teams to keep themselves synched.

1. When you name a tag that you believe should be added as a standard tag, you should be able to identify it. We recommend “locking” these tags. Later we can pull the tags that were “locked” and work on creating “real” IEC tag names.
2. Create all tag names on the master computer; then copy the customtags.xml file (located in the OS-specific **data** directory: Data/Profile Editor/ Vx.x) to each computer that will be used to create and edit device-types/profiles.
3. When creating a new tag name, be sure to complete all of the alarm/logging filter (metadata) data fields. Without these fields properly completed, the real-time and alarm filters, and future reports/analysis will not work.

It is often easier to “copy from” a similar tag when creating a new tag. This will pre-populate all of the fields, including the metadata/filter data. Then you can change the tag as needed.

## How should we manage categories and subcategories?

We recommend that each integration team decide in advance which categories and subcategories they will use. The automation interface requires the entire Profile name (which uses the category and subcategory as part of its name). Thus, you must be consistent in naming if the profiles are going to be shared and re-used.

1. Category should be used for a vendor.
2. Subcategory should be used to describe a type of device.
3. From the master computer that has the Profile Editor installed, create the categories and subcategories that you plan to use.
4. Copy the DeviceTypeCategories.xml file (located in the OS-specific **data** directory: Data/Profile Editor/ Vx.x ) to every computer being used to create profiles.

## How should we manage units and unit templates?

Until the Profile Editor has been expanded to include an import/export utility it will take work on part of the integration teams to keep themselves synced.

Create all units and unit templates on the master computer. Then copy the units.xml and unitstemplates.xml (located in the OS specific **data** directory: Data/Profile Editor/ Vx.x ) to each computer that will be used to create/edit device-types/profiles.

## When should I create a device type rather than device profile?

Create a new device type, instead of a profile, when the addressing for a specific tag needs to change. An example:

The integration team can choose which Input to which they will wire circuit breaker status and position. In this case, the tags for circuit breaker status and position would have different addressing, based on how that particular circuit breaker is wired. We recommend a new device type in this case.


© 2011 Schneider Electric All Rights Reserved

## How do we share or transfer device types and device profiles between computers?

Until the Profile editor has been expanded to include an import/export utility, it will take work on part of the integration teams to keep themselves synced.

**You must ensure that you have maintained synchronization** between all the computers for:

- units and unit templates
- categories and subcategories
- custom tag names

**To transfer or share device types**:

1. On the source PC: From the OS-specific Data/Profile Editor/ Vx.x directory, locate the device type of interest (<Device Type name>.xml). Copy it to the corresponding directory on the destination PC.
2. On the source PC: From the Data/Profile Editor/ Vx.x /DeviceTypes directory, locate the device type of interest (<Device Type name>.xml). Copy it to the corresponding directory on the destination PC. (You must locate the proper version of the Profile Editor. It will be associated with the version of the Profile Editor that was used to create it.)

   If you do not know the profile name, or if you want to pull over all of the profiles for a given device type):

   a. From the Profile Editor, Add/Edit Device Type screen, click “Edit Existing.”
   b. Choose a Device Type to Create From.
   c. Click “Display Associated Profiles.” Note the list of associated profiles to the chosen device type.
   d. Copy the profile file(s) (<Device Profile name>.xml) to the corresponding directory on the destination machine.

**Importing files into the Profile Editor**

Use the import feature to import tags, device types, and profiles into the Profile Editor. You can import either an existing project or SCL files. See *Importing Files into the Profile Editor* on page 72 for full details.

## How can we re-use previous Profile Editor projects?

Until the Profile editor has been expanded to include an import/export utility it will take work on part of the integration teams to keep themselves synced. It is recommended that you

**You must ensure that you have maintained synchronization** between all the computers for:

- units and unit templates
- categories and subcategories
- custom tag names
- device types
- device profiles

**To transfer/share projects**:

© 2011 Schneider Electric All Rights Reserved

1. On the source PC: From the OS-specific Data/Profile Editor/ Vx.x directory, locate the project of interest. Copy the entire project-specific directory to the corresponding directory on the destination PC.
2. On the source PC: From the Data/Profile Editor/ Vx.x /DeviceTypes directory, locate the project of interest. Copy it to the corresponding directory on the destination PC. (You must locate the proper version of the Profile Editor. It will be associated with the version of the Profile Editor that was used to create it.)

**Importing files into the Profile Editor**

Use the import feature to import tags, device types, and profiles into the Profile Editor. You can import either an existing project or SCL files. See *Importing Files into the Profile Editor* on page 72 for full details.

## How do we synchronize a new PC with the master Profile Editor PC?

To synchronize a new machine with the latest device types and profiles from your master Profile Editor PC, you can:

- Use the Import feature to import tags, device types, and profiles from either an existing project or from SCL files. See *Importing Files into the Profile Editor* on page 72 for full details.
- On the source PC: From the OS-specific Data/Profile Editor/ Vx.x directory, copy the entire OS-specific Data/Profile Editor/ Vx.x directory to the corresponding directory on the destination machine.

## What do I do before I add or remove devices in the Profile Wizard?

1. You must close all open dbf files in the Project Editor (or in Excel).
2. If you are removing a device, check “pack database after removal” on the last page of the wizard.
3. Any changes that you made inside the Citect Project Editor (such as setpoints or data type modifications) are lost when you delete the device from PowerLogic SCADA.

## What are the requirements for device names?

**Device Name**:

Keep Device name ≤ 16 characters. Use _ as a separator.

If you use a naming convention that incorporates location, you will be able to do filtering on alarm location, etc.

- For example, Site_Building_Panel_device would be named Sx_Bx_Px_Device. (Site1_Building1_Panel1_CM41 — S1_B1_P1_CM41).
- The fewer levels you have, the more characters you can have in each level.

**Device Comment**:

Use this field for an alias of the device name.

This comment will be placed in the Equipment database, which is accessible from Cicode.


© 2011 Schneider Electric All Rights Reserved

## How do I troubleshoot device communications issues?

PowerLogic SCADA drivers provide default communication settings that work with most devices. However, in cases when communication losses occur in PowerLogic SCADA, use this checklist for finding the issues.

1. Initial checks, if the device is attached via a gateway:
   - Ensure that all communication settings are correct on the gateway and device.
   - Check the gateway timeout. A setting that is too low will cause many timeouts to occur. A setting that is too high will impact performance. We recommend a 3 second timeout, because most devices work well with this setting. Some devices may require a higher timeout (5 seconds).
2. In all communication setups (also see the driver help for parameters):
   - Ensure that the PowerLogic SCADA driver timeout is correct. We recommend that you set this to:

     gateway timeout x number of clients + padding

     Example: If the gateway timeout is 3 seconds and there are 3 clients, set the timeout in PowerLogic SCADA to 10 seconds.
   - Check the maximum block read size. Some devices do not handle large block reads well. When you lower the maximum block read size, the requests are smaller and faster. The downside is that more requests will be sent to the device, and tags will refresh more slowly.
   - Check the device to see if there are registers that it cannot read. Some devices do not allow access to all registers.

     Example: Data is in register 100-125 and 130-150. PowerLogic SCADA will perform one read from 100-150. If 126-129 do not allow reading, this packet will return an exception. Use the appropriate logic code to mark these registers as invalid reads.
   - If there are still timeout/no response issues, enable retries on exception. Some devices may not respond if they are performing other functions. In this case, a0x0A or 0x0B exception will be returned to PowerLoic SCADA, which will cause a communication loss. Enabling the "retry on exception" will re-try the request.

## How do I use MODBUS communications methods?

MODBUS TCP/IP via Gateway: Use this for any device that is not speaking TCP/IP natively. These devices connect through a gateway device such as an EGX or ECC.

MODBUS TCPIP: Use this for any device that can speak TCP/IP natively. This includes CM4 or PM8 devices that have an ECC card installed.

## How can I add more than one device at a time?

The Automation Interface requires that the profiles have already been exported from the Profile Editor to the PowerLogic SCADA project.

If the automation spreadsheet attempts to add a device that is already present in the project, an error will be thrown.

In the event that an error is thrown (for invalid profiles, communication parameters, etc), the row containing the error will display in Excel. To prevent duplicate device

© 2011 Schneider Electric All Rights Reserved

entries from being attempted, you must remove any rows above the row indicated in the error message.

If you need to keep a record of the devices added to the system, then keep each of the spreadsheets that was used to install devices in a known location for that customer.

The Setup Sheet needs to be modified for each project. Specify the entire path for each file.

The Input Sheet requires the following:

The entire path name for each profile. The path name for a profile is based on the category and subcategory from the Profile editor.

Example: Schneider Electric.Monitoring Device.Branch Circuit Monitor Full

## In PowerLogic SCADA, what are the naming conventions for servers and clusters?

There is no enforced naming convention for server and cluster names, other than the restriction that each server name and cluster name must be unique. Cluster names must be a maximum of 16 characters, no spaces; they may not begin with a number.

Each team should come up with a naming convention for the servers and clusters. If the team is consistent, it will make it easier to edit/create the automation spreadsheet used for device addition.

## How and when do I create users for the Runtime environment?

New projects do not have any users created by default.

The default PowerLogic SCADA graphics objects (such as circuit breakers and alarm pages) are constructed using a pre-defined set of user privileges the PowerLogic SCADA security grid). During development, you must have users of various privilege levels for testing purposes. Create users for each of the various levels according to the PowerLogic SCADA security grid. To make the best use of these privileges, we recommend that you use this security grid when adding users as you create new projects.

See *Default User Access Settings (Privileges)* on page 159. For additional information, see Using_Security.html in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerLogic SCADA 7.20 > PowerLogic SCADA Help).

## How do I manage projects in the Citect Explorer environment of PowerLogic SCADA?

Although the designer may wish to organise each project in a particular way to suit customer's needs, the following is a recommended best practice:

1. Keep original ‘Master’ copies of the PLS_Example and the PLS_Include projects for reference.
2. The Services group may develop a group-wide “include” project that will act as a conduit between the PLS_Include project and all customer projects (e.g., "Group_Include"). This will make the upgrading of PLS_Include much easier, as it will be the only project that must be modified to be compatible with the new version in the group-wide include project.

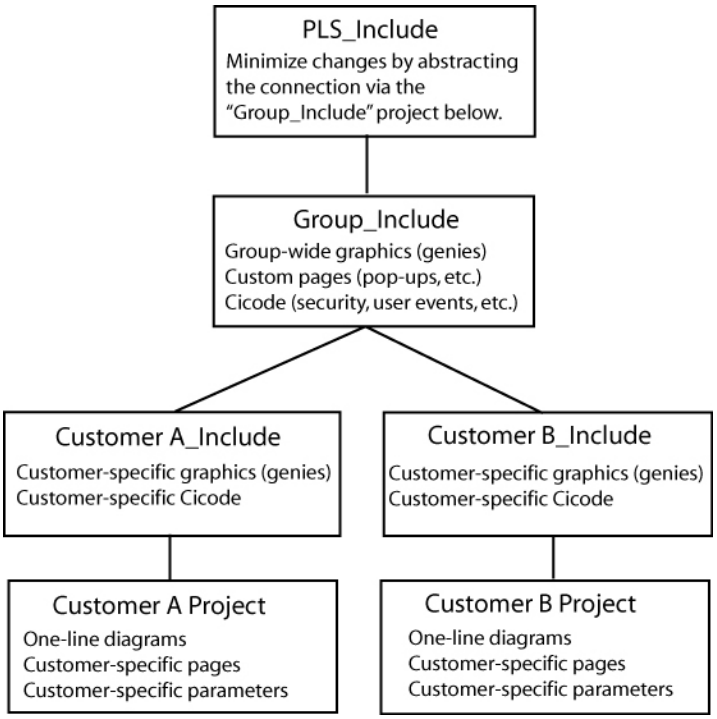

© 2011 Schneider Electric All Rights Reserved

Any changes made to the PLS_Include project should be made at the Group_ Include project level. This would involve removing portions of the code from the PLS_Include project, modifying the code and saving it in the Group_Include project. By removing (or commenting out) the original code and placing the new code in the Group_Include project, a layer of abstraction is preserved, further simplifying the upgrade process. In other words, the only changes to PLS_Include should be code removal.

3. When a new customer project is started, also create a customer-level “include” project.

   Always back up and restore the customer project and its associated include projects together.

   Always restore include project(s) before restoring the customer (or top-level) project.




4. Upgrading PLS_Include:

   Document all changes to PLS_Include. This is absolutely necessary when upgrading to a new version of the PLS_Include project.

   Minimize changes to the PLS_include project.

   Abstract as many changes to the PLS_Include project as possible. This should be accomplished by the use of multiple include projects as shown in the diagram above.

   New versions of PLS_Include will include a detailed description of each change, allowing you to merge the old and new versions.

   New versions of PLS_Include will maintain backward compatibility where possible.

© 2011 Schneider Electric All Rights Reserved

## On the Graphics page, what do I need to know about creating genies?

1. Creating a new genie:

   The easiest way to create a new genie is to use an existing genie from the library. This ensures that the new genie is compatible with the PowerLogic SCADA system, and that it preserves these features:

   — Compatibility with animated busbar logic is maintained. This gives you the ability to use the setting from the Color Change Tool for animated busbar colour.

   — A sizing guide (a dotted rectangle) is included; it displays during graphics edit mode. This guide ensures that new genies can be swapped with existing genies without the need to recreate portions of the drawing.Save the new genie in the appropriate project (do not overwrite the provided genies).

   Save the new genie in the appropriate project (do not overwrite the provided genies).

2. Copying a genie to another project:

   Open the genie in the graphics editor, and do a <save as> into another project/library.

3. Genie Form Files:

   Any new genie (copied or created) will not have a .frm file entry associated with it. While the new genie is functional, it will show a cryptic unformatted properties box in the Graphics Editor. You can create your own .frm file with the needed entries by following the instructions available in the Citect Knowledge base.

   If you want to use the .frm dialog box that belongs to the genie you copied, go to the PLS_Include library; locate the .ctm and .ftm files. Each library has its own .ctm/.ftm files that include the description for every genie in the library. (This is an ascii text file that you can open in any text editor.) Find the genie that you copied (or on which you're basing the new form). Copy the portion that matches the copied genie, and create a .frm file that has the desired library name on it. Copy in the text from the .frm file. Restart Citect Explorer or it will not detect the new .frm.

4. Genie Sizing

   The provided genies come in two sizes: size 1 and size 2. When making a new genie for reuse among multiple projects, it will be beneficial to create a genie for both sizes. Follow the same steps for both sizes (sizing guides are provided for both sizes).

## How do we customize existing templates?

Template Editing:

All objects on the page contain one or more Animation Numbers (ANs). Symbols take one AN while genies may take tens to hundreds of ANs. Placeholder ANs allow you to add objects to a template that is used on existing pages.

Some default templates contain ANs that have associated Cicode functions that rely on the animation number to remain a fixed number. For this reason, we have pre-allocated a set of ANs for the default templates. The base normal template uses ANs 1–263, and it has placeholder ANs from 264–500. When customizing this template, you should use the placeholder ANs as required.

You can place an AN (or a placeholder AN) on the page by using the “System 3.x/4.x tools available in the Graphics Builder under Tools< Options.

The default template uses ANs 1–263 and it has placeholder ANs from 264–500.


© 2011 Schneider Electric All Rights Reserved

New objects added to a page or template will take the next available ANs. Any previously used (and now abandoned) ANs will be reused.

To add an object on the template, you must:

1. Open the template.
2. View the page properties and change the page width to 2000. This will reveal the hidden placeholder ANs on the page. You may have to change the width to a wider dimension for widescreen templates.
3. Determine how many ANs the new object requires. (You can place the new object on a blank page and then view the object in the object browser.)
4. Remove exactly the amount of ANs to allow the new object to be placed on the template. Remove ANs beginning with the lowest available placeholder AN (in the default template, this would be 264).
5. Place the object on the template.
6. Save the template.
7. Create a new page based on this template.

8. Drop a numeric object on the page.

   a. This object's AN should be 502 (501 is reserved for placing the template on the page).
   b. If the object has an AN less than 502 then you have unused AN(s) on the template. This must be resolved. (Place additional ANs on the template to rectify this situation.)
   c. If the object has an AN greater than 502 then you have too many ANs on the template (a AN on the template is going beyond the 500 limit). You must find the culprit (via the object browser) and rectify the situation using the steps above.

## How do I change the default pickup/dropout text for alarms?

The default 'pickup/dropout' text is shown as Appearance/Disappearance.

To change globally:

This text may be changed by configuring INI parameters in the citect.ini file. For more information, see the System Integrator's Guide (*Graphics Library Parameters* on page 193)

This is the *global fallback text* that will be used in the event that pickup/dropout text is not specific on a per-alarm basis in the Device Profile. You can specify the per-alarm pickup/drop-out text on the profile tab in the Profile Editor.

To change on an individual basis:

See the System Integrator's Guide (*Viewing Device Profiles* on page 51, "Alarm On Text" and "Alarm Off Text").

## What can I modify during runtime?

**Set-point Configuration during runtime**: In the Alarm Server entry in the project editor, view the advanced properties (F2) and set publish alarm properties = TRUE

**Events Accessible at runtime**: By default, 5000 events are accessible (view, filter, print) during runtime. Additional events are logged to disk. The number of events

© 2011 Schneider Electric All Rights Reserved

accessible at runtime may be changed by the [Alarm] SummaryLength parameter in the INI file.

## Why do the browser navigation buttons not work?

If the browser navigation buttons do not work when you are viewing the runtime window, you have probably added a new page, but have not done the following:

- added the startup page to the Page parameter. See *Setting a New Page as the Project Startup Page* on page 118 for help.
- left the INI settings at <default>. In the Computer Setup Wizard, General Options Setup screen, do not change the StartupPage field; leave it as <default>.

## What can I set up in logging and archiving?

**Event Logging and Archiving**:

Event fields that are logged to disk may be configured by adjusting the AlarmFormat parameter.

There is no automatic maintenance performed on the log files. It is important that the log/waveform data be cleared out periodically (to prevent the hard drive from filling up; this does not affect performance).

## On the Graphics page, how are colour changes displayed for voltage levels?

Definitions:

**Voltage Level**: This is a number from 0–3 that relates to the selected “ACTIVE” colour in the Color Change Tool. This is a graphics-level number that is used for animation purposes (in expressions and Cicode).

**Nominal Voltage**: This is the actual voltage present on the line at the customer’s site.

1. Before creating the first graphics page of a new project, determine the nominal voltages that will be assigned to voltage levels 0–3. Examine the customer’s power system to determine nominal voltages required.
2. As a team, decide which voltage level will represent the highest nominal voltage present at a customer’s site (0 or 3). This will provide for easier code re-use and project management.
3. Document the nominal voltages and voltage levels for each project, and keep them with the project (in a text file in the project directory or in a comment in a Cicode file).
4. Run the Color Change tool after any changes to graphics and at the very end of the development process. This ensures that all graphics have the same colour configuration.

   Always update pages after running the Color Change Tool.

   The Color Change Tool will change colours on all projects that are included by the top-level project. Care must be taken to not ‘colour change’ a project unintentionally.

   Example Project Documentation:


© 2011 Schneider Electric All Rights Reserved

| Colour Change Voltage Level | Nominal Voltages | Colour |
|---|---|---|
| 0 | 240 | blue |
| 1 | 480 | green |
| 2 | 4160 | purple |
| 3 | 12470 | red |

## How do I create and configure busbars?

When drawing one-line diagrams:

Analyse the drawings at a customer site.

Number the busbars consistently on the one line diagram(s). If busbar 14 spans across multiple pages, it should be numbered busbar 14 on all pages. Label the voltage level (0–3) on each busbar.

Uses for Line Active:

**Page Connections**: Many one-line diagrams will span multiple pages in PowerLogic SCADA. To connect these pages together, you must use the line active field of the 'incomers' of the second and subsequent pages. Set the line active field of the incoming busbars on these pages to an expression that references the nearest device on the same busbar of the previous page.

**Metered Busbar**: Many busbars are actually metered. It is more accurate to allow these metering devices to dictate state than to rely solely on the simulation (see Expressions below).

Configuration of Line Active:

**Simulation**: If the Line-Active field is left blank, the busbar state will be determined by surrounding devices.

**Expressions**:

A Cicode expression in the form of Device\Tag > Nominal Voltage (I.E., S1_B1_P1_CM41\MMXU1\PhV\zavg > 120).

If the expression is TRUE, the ACTIVE colour will be shown. The active colour is determined by the voltage level assigned.

If the expression is FALSE, the DE-ENERGIZED colour will be shown.

**Hard-Coded**:

If no upstream devices are available (in the event of an incomer, for example), you may have no other choice than to 'hard code' this field to a '1'. This forces the busbar to always be ACTIVE.

## What INI parameters should I use for debugging?

We recommend that you contact Technical Support before performing any debugging.

**Parameter: [PowerLogicCore]**

DebugCategory = Error

DebugLevel = All

LogArchive = 1

LogFileSize = 200000

© 2011 Schneider Electric All Rights Reserved

Parameter Details:

**DebugCategory** defines which message categories to log. (See table below).

**DebugLevel** defines debug levels of messages to be logged. (See table below).

**LogFileSize** specifies maximum log file size in kilobytes. After reaching the limit, file is saved with suffix ".BAK" and new file is started over.

**Debug Levels**

The following debug levels are accepted by PowerLogic driver core library:

WARN: log all warning level messages

ERROR: log all error messages

TRACE: log all trace messages

DEBUG: log all debug messages

ALL: include all level messages

**Debug Categories**

PowerLogic core library and drivers messages are grouped in categories. Each of these categories can be enabled independently from others in any combination.

MISC: miscellaneous messages that do not all into any other category

ALARM: messages related to alarms, regarding collection and detection

WAVE: messages related to waveforms -- waveforms download, processing

REAL: messages related to real-time data collection

STATE: messages related to internal object-state changes

CORE: core events that do not fall into driver-specific logic

MODBUS: TCP/MODBUS messages

ENTRY: trace messages produced when driver API entry points are called by PowerLogic SCADA

PORT: traces related to the port events

UNIT: traces related to specific unit events

WAVETOALARM: not used

DATAPOINT: debug messages related to data points

STATISTICS: enables driver statistics data output

COMMAND: messages related to commands

REPLICATION: messages produced by replication subsystem

ALL: enables all categories

**Parameter: [Debug]**

Menu = 1

Parameter Details:

The Menu parameter determines whether the Kernel option is displayed on the control menu of the PowerLogic SCADA runtime menu. This can also be enabled using the Computer Setup Editor.

## How do I tune my system for best performance?

There are several parameters that you can use to enhance your system's performance:

**Driver-tuning parameters**:

**Parameter** (Back Polling Rate): **[SEPAM40]**


© 2011 Schneider Electric All Rights Reserved

CacheRefreshTime = 1000

InitUniCheckTime = 120

Retry = 3

Timeout = 1000

**Parameter Details**:

The CacheRefreshTime parameter controls the maximum rate at which the driver will attempt to repopulate its cache. If the driver cannot refresh its cache within the time period specified, it will collect data as fast as the network allows.

This back polling rate can be global to all devices or tuned up to a specific I/O device.

The InitUniCheckTime parameter controls how long the driver will wait before attempting to bring a device online after it has gone offline. This value can be decreased to bring offline devices back into service in a shorter period of time. In a multi-drop scenario, this time should be relatively long, to prevent init unit requests from stalling communications to the rest of the devices on that port.

The Retry parameter defines the number of retry attempts for specific MODBUS requests. Retries will only occur in response to the MODBUS errors which are defined below.

The Timeout parameter controls how long the driver will wait for a response from a device before setting that device as offline. This value should be greater than the device/gateway timeout period.

**Parameter: [Device]**

WatchTime = 5000

**Parameter Details**:

**Device WatchTime** is the frequency that PowerLogicSCADA checks devices for history files and flushes logging data to disk.

Default: 5000

Range: 1000–3600000 milliseconds.

**Miscellaneous Parameters**

**Parameter: [Kernel]**

Task = 20000

**Parameter Details**:

Kernel Task is the number of tasks. Increasing the number of kernel tasks is used when “Out of Kernel Task” message is received. The change will be likely for large systems.

Default Value: 256

Range: 50–32767

**Parameter: [Page]**

ScanTime = 250

**Parameter Details**:

Page ScanTime determines how often the Animator refreshes a graphics page at runtime.

Default: 250

Range: 1–60000 milliseconds

**Parameter: [ALARM]**

ScanTime = 500

© 2011 Schneider Electric All Rights Reserved 

**Parameter Details**:

**Alarm ScanTime** determines the rate at which alarms are scanned and processed.

Default: 500

Range: 0–60000 milliseconds

## If a tag is configured in PowerLogic SCADA, how is it polled in the device?

In other words, is a tag only polled on demand when it is requested by a client (for example, when the operator displays a page with the tag on it)? Or are all configured tags polled all of the time, with the relative polling rates/communications bandwidth carefully managed?

The ModNet driver polls real-time tags on a user demand basis (when a user opens a page with the tags on it). Therefore, the time to retrieve data will vary, depending not only on the communications bandwidth, but on the amount of data being requested. This can vary significantly, depending on which pages are displayed by the operators at any particular time.

The PWRMODBUS driver polls all configured tags; however, different types of tags can be polled at different relative rates, and the available communications bandwidth is carefully managed. This approach means that tag update rates are not subject to the scaleability issues associated with operator actions (as is the case for the ModNet driver). It is also advantageous in that performance issues associated with communications bandwidth or IODevice response times can be determined at SAT/time of implementation and are not subject to significant change during operation.

The different tag types can be allocated relative importance in data requests, expressed as a percentage. (See Bandwidth Allocation Parameters in *Performance Tuning Parameters* on page 180.) Keep in mind that any unused bandwidth allocation (from, for example, events retrieval) is made available for other data types to use. If the event does not need the default 25% allocation, it will be made available to the other parameters (real-time tag retrieval, etc). This potentially increases the update rate of real-time tags.

Additionally, the real-time tag relative scan rate based on priority can be set to three different levels. (See Tag Scan Rate Parameters in *Performance Tuning Parameters* on page 180.) This means that, if some real-time tags are more important that others, you can set their relative priorities. For example, configuration tags vs. important real-time tags vs. normal real-time tags.

## Device popup from a one-line: Why do the fields overlap?

This is controlled by a parameter entry:

Section: Page

Name: EquipDetailDescLength (the total number of characters in a single row of this popup)

Default = 48. The problem will occur with a larger font or if the window is resized. The default value of 48 can be changed or the window and associated genies can be resized.


© 2011 Schneider Electric All Rights Reserved

## Can I change the %CLUSTER% name in the Profile Wizard?

No. If you change the placeholder %CLUSTER% to any other name in the Profile Wizard, the system will be unable to find the actual cluster to which it refers.

## When viewing PowerLogic SCADA from a web client, I see a licensing error. How do I fix it?

First, verify that the appropriate licensing is available.

A web client requires the same number of ports open as a standard control or view-only client. You only need to allow communication to those servers from which the client needs information. For example, alarm, trend, reports, and I/O.

When you open PowerLogic SCADA from a client computer, and you see a licensing error, make these port settings. This will open both PowerLogic SCADA and the license key server (7001: USB sentinel key port) in the routers and firewalls between the client and the server. The ports that you need to open are listed in **boldface**:

| Default Port | Server Type | Server Role |
|---|---|---|
| 21 | FTP server | page downloads for IDC |
| **80** | **Web server** | **project files for Web client** |
| 2073 | CTAPI | CTAPI communications |
| 2074 | client | Cicode debugging |
| **2075** | Reports server | Reports server communications |
| **2076** | **Alarm server** | **Alarm server communications** |
| **2077** | **Trends server** | **Trends server communications** |
| **2078** | **IO server** | **legacy I/O communications** |
| 2079 | IDC | Internet display server/client communications |
| **2080** | **Alarm server** | **alarm properties connector** |
| 2081 | time server | time server communications |
| **2082** | **IO server** | **publish subscribe I/O server communications** |
| 20222 | ODBC | ODBC server |

## A device can prevent writes to its registers: how do I ensure that writes are successful?

PowerLogic SCADA cannot provide feedback about whether a write to a device register is successful. If a device is capable is preventing (blocking) writes to its registers (for example, Sepam), you need to verify that its “block” feature is not enabled. Do this at the device.

NOTE: In Cicode, you can also use the tagwrite function in blocking mode, i.e., bSync parameter = true; Check the return code: 0 = success, anything else = error. For more information, see the Cicode Programming Reference help file.

© 2011 Schneider Electric All Rights Reserved

## How do I prevent PowerLogic SCADA from accidentally making invalid areas in memory available to reads and writes?

PowerLogic SCADA normally optimises its packets for greatest performance. This optimisation can sometimes includes invalid areas of memory in devices. These invalid areas can be specifically defined and excluded from optimisation packets created by PowerLogic SCADA. See *Advanced Tag Blocking Capabilities* in *Performance Tuning Parameters* on page 180 for more information.

## How do I create an audit in the Event Log for user logins and logouts?

```
//LOGOUT


FUNCTION
PLSLoginUser()


    //INT iPage = PageInfo(1);
    INT iPage = WinNumber();
    IF mbLoginFormShown[iPage] = TRUE THEN
    RETURN; //form already shown
    END
//prevent multiple forms
mbLoginFormShown[iPage] = TRUE;
IF (UserInfo(0) <> "0") THEN
        // Confirm User Action
        IF (0 = Message(StrToLocalText("@(Confirm)"), StrToLocalText("@(Logout)"),
        1+32)) THEN
        PLSAlmDspEventAdd(0, 0, 1, "User Logout", Name(), "Logout", "");
        Logout();
        END
        mbLoginFormShown[iPage] = FALSE;
        RETURN;
    END
    IF (0 = LoginForm())
    PLSAlmDspEventAdd(0, 0, 1, "User Login", Name(), "Login", "");
    END
    mbLoginFormShown[iPage] = FALSE;
END
```


© 2011 Schneider Electric All Rights Reserved

## Why do I get an "out of memory" message when I try to run the Automation Interface?

The problem is that you did not have Microsoft Excel installed before you installed PowerLogic SCADA. The only solution is to uninstall both Excel and PowerLogic SCADA; then, reinstall first Excel and then PowerLogic SCADA.

## Why am I seeing #COM for circuit breaker status in the genie status page?

If this is a Micrologic P device, and it does not have a CCM, you will not be able to view data referring to circuit breaker status, e.g. racked in/racked out. When there is no CCM, the device profile should not have tags that refer to the CCM.

## Why can't I acquire waveforms in the waveform viewer?

The "acquire" feature (the "A" button on the waveform viewer) does not work in PowerLogic SCADA. You can, however, view waveforms from device onboard waveform files. To do this:

- At the device or in the meter configuration software, add the appropriate alarm, and enable automatic capture of the waveform when the alarm occurs.
- In the Profile Editor (Create Device Profiles tab), check the Waveform box for the alarm you added.
- When the alarm occurs, the waveform is captured. You can view the waveform in the Alarm Log. You can also view alarms/waveforms from a drawing in the runtime environment. Click the genie for the device; right-click the alarm to view the waveform.

  Note that, in very large systems, it could take as much as an hour for the waveform to appear.

## Why won't the Excel DBF Add-In toolbar install?

When you are installing the Excel DBF Add-In toolbar, you may see this error: "Error 1308. Source file not found....."

You can click "ignore" at this error, and the install will finish. The next time you open Excel, the DBF toolbar will display.

## What causes the "First dbf record" error message? How do I keep it from happening?

The error message "First dbf record" tells you that a project is not found. This happens when you add a project, and then rename it or delete it. Then, when you try to create a new project, you see this error message.

To resolve this issue, simply shut down and then restart the Citect Explorer.

© 2011 Schneider Electric All Rights Reserved

# Index of Terms




© 2011 Schneider Electric All Rights Reserved

**D**




© 2011 Schneider Electric All Rights Reserved

## E



## F



© 2011 Schneider Electric All Rights Reserved

**G**



**H**



**I**




© 2011 Schneider Electric All Rights Reserved

**L**



**M**



**N**



**O**



**P**



© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved

© 2011 Schneider Electric All Rights Reserved

**U**



**V**



**W**



**X**




© 2011 Schneider Electric All Rights Reserved

PowerLogic™ SCADA System Integrator's Manual
User Guide

Schneider Electric
295 Tech Park Drive, Suite 100
LaVergne, TN 37086 USA

For technical support:
Global-PMC-Tech-support@schneider-electric.com
(00) + 1 250 544 3010
Contact your local Schneider Electric sales representative for assistance or go to www.schneider-electric.com

PowerLogic and Citect are trademarks of Schneider Electric. Other trademarks are the property of their respective owners.

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

63220-100-200E1 06/2011
Replaces 63220-100-200D1, dated 05/2011
© Schneider Electric All Rights Reserved.