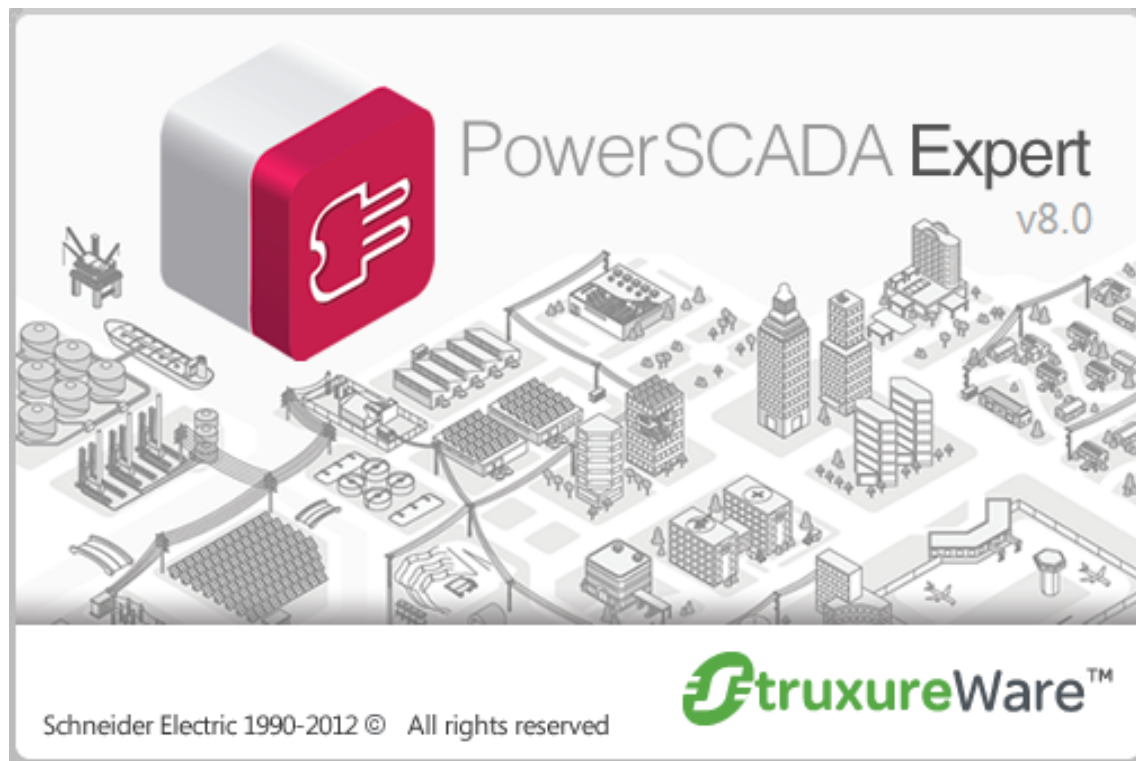


PowerSCADA Expert v. 8.0 System Integrator's Manual

User Guide

63220-100-200I1

03/2015



Safety information

Important information

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



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



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

DANGER

DANGER indicates 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, **could result in** death or serious injury.

CAUTION

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

NOTICE

NOTICE is used to address practices not related to physical injury. The safety alert symbol shall not be used with this signal word.

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.

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

Safety precautions

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



WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not use the software for critical control or protection applications where human or equipment safety relies on the operation of the control action.
- Do not use the software to control time-critical functions because communication delays can occur between the time a control is initiated and when that action is applied.
- Do not use the software to control remote equipment without securing it with an authorized access level, and without including a status object to provide feedback about the status of the control operation.

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



WARNING

INACCURATE DATA RESULTS

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

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

Contents

Safety information	1
Important information	1
Please note	1
Safety precautions	1
Contents	i
Introduction	1
PowerSCADA Expert—A Complete Solution	1
Assumptions	1
What is SCADA?	1
What is PowerSCADA Expert?	2
Prepare for PowerSCADA Expert	2
Components of a Project	2
Configuration Tools	2
Runtime Environment	3
Supported Device Types and Protocols	3
How Do Drivers Work?	3
Two Subscription Types	3
Subscription Expirations	4
Edit Driver Parameters	4
Problems with Duplicate Devices using PowerLogic Drivers	4
System Requirements and Installation Instructions	5
Uninstall and Reinstall PowerSCADA Expert	5
Quick Start: Typical Workflow	5
IEC 61850 System Setup Workflow	8
Get More Information	9
Manuals	9
Help files	9
Customer support	9
Use the Migration Utility	10
The Profile Editor	12
Overview of the Profile Editor	13
Typical Workflow Illustration	13
Workflow Overview	14
Create/Edit Device Type	15
Create/Edit Device Profile	16
Create/Edit Unit Templates	17
Launch the Profile Editor	17
Locked and Custom Icons	17
Set the Screen Resolution	18
Profile Editor Main Menu Options	18
Add Engineering Unit Templates, Units, and Conversions	19
Set Up Engineering Templates and Select Conversions	19
Apply Conversions	21
Delete a Template	21
Add or Edit a Base Engineering Unit or Conversion	22
Edit a Base Engineering Unit or Conversion	23
Delete a Base Engineering Unit or Conversion	23
Define Device Types and Tags	23
Device Type Screens and Workflow	23
Use the Define Device Type Tags Tab	24
Add, Edit, or Delete a Device Type	26
Edit a Device Type	27
Delete a Device Type	28

Assign Tags to Generic I/O Points	28
Print the .CSV File	29
Set Up Device Type Categories	29
Add a Category or Subcategory	30
Edit a Category/Subcategory Name	30
Delete a Category or Subcategory	30
Edit Functional Addresses	31
Add Custom Tags and Tag Addresses	31
Set Up Custom Tags	31
Edit a Custom Tag	33
Delete a Custom Tag	34
Edit Tag Addresses	34
Real-Time Tag Addresses	34
Onboard Alarm Tag Addresses	36
Reset Tag Addresses	36
Control Tag Addresses	37
Edit Address Information	38
Add a New Tag Address	38
Edit Generic Tag Addresses	39
Create Device Profiles	39
Enable Waveforms	39
View Device Profiles	39
Add Edit or Delete Device Profile	41
Add a Profile	41
Edit a Profile	42
Delete a Profile	43
Select Trend Intervals	43
Set Up Trend Intervals	43
To add a trend interval:	43
To edit a trend interval:	43
To delete a trend interval:	44
Edit IEC 61850 Datasets	44
Create a New DataSet	44
Create a DataSet from an Existing DataSet	44
Copy a DataSet to a Device Type	45
Edit and Delete DataSets	45
Edit IEC 61850 Report Control Blocks	45
Create a New Report Control Block	45
Create a Report Control Block from an Existing Report Control Block	46
Copy a Report Control Block to a Device Type	46
Edit and Delete Report Control Blocks	46
Create a Composite Device Type	46
Create Data Concentrator Device	48
G3200 Device Setup	48
For use with multiple devices	49
In the Profile Editor	49
In CET850	49
In PowerSCADA Expert	49
For use with a single device	49
In the Profile Editor	49
In CET850	49
In PowerSCADA Expert	50
DNP3 Protocol Support	50
Set Up Projects in the Profile Editor	50
Project Screens and Workflow	50
Typical Workflow	51
The Set Up Projects Tab	51

Adding a Project	52
Edit a Project	53
Delete a Project	53
Edit and Delete Information in a Project	54
Customize Tag Names	54
Add Project Parameters	54
Import and Export Project Files	54
Export a Project	55
Before you export	55
To export a Profile Editor project to the PowerSCADA Expert project:	55
To Move Files if the Profile Editor is not with the Server (Reuse of a Project)	56
Profile Editor Export	56
SCL Export	57
Exporting the File	57
Import Files into the Profile Editor	57
Import SCL Files	59
Import Filter Screen	59
Functional Constraints	59
Report Control Blocks	59
Import Reconciliation Screen	60
Left-hand pane:	61
Middle pane:	62
Right-hand pane:	63
Complete the Import	63
Using Import Templates	64
Creating a New Template During Import	64
Applying a Template During Import	64
Deleting a Template	65
Tag Types	65
IEC 61850 Tag Construction	65
Define an Enumeration	66
Use Special Tags to Control Circuit Breaker Status	67
Format Code Definitions	67
Real-Time Format Code Definitions	67
Alarm Format Code Definitions	68
Control Format Code Definitions Rules of Operation	69
Predefined Control Format Codes	69
Predefined Reset Format Codes	69
Custom Control and Reset Format Codes	69
Logic Codes	69
Block Writes	85
The Profile Wizard	87
Before You Use the Profile Wizard	87
Add the Cluster, Network Address, and Servers	88
When You Set Up More than two I/O Servers per Cluster	90
Enter Copy for Translation	91
Using the Wizard to Add and Remove Devices	91
Add a Device to a Project	92
Add a TCP Device to a Project	92
Add the TCP Device	92
Add a Serial Device to a Project	94
Adding the Serial Device	94
Add a DNP3_TCP Device	96
Add the DNP3_TCP Device	96
Add an IEC 61850 Device to a Project	98
Add the IEC 61850 Device	98
LDName	99

BRCBs and URCBs	99
Port Names	100
Edit Devices in a PowerSCADA Expert Project	100
Edit a Profile and Add it Back to the Project	100
Edit a Device in PowerSCADA Expert Only	101
Add Device Data in PowerSCADA Expert Only	101
Remove a Device From the Project	101
Add Multiple Devices through the Automation Interface	102
Use the OLE Automation Interface to Add and Remove Devices	103
Examples	104
C#	104
Excel VBA	105
Command Line Interface	105
Interface Workflow	105
Prerequisite information	106
Interface	107
The Design Time Environment	109
Typical Workflow	109
The PowerSCADA Expert Pages	110
Create a Project	110
Reuse Projects Created in the Profile Editor	111
Before You Begin	111
Add a New Graphics Page	111
Change the Background Color of Pages	112
Change the Genie Color in Project Pages	112
Set a New Page as the Project Startup Page	112
Create a One-Line on a Graphics Page	112
Enable Lockout/Tagout	114
Use Menu Configuration to Edit Pagemenu.dbf (Change the Graphics Page Appearance)	115
Create New Genies	116
To create a copy of a genie:	117
To create a custom symbol for a custom genie:	117
Set Up IEC 61850 Advanced Control	117
Enable the Advanced Control	118
Delete Information from PowerSCADA Expert	118
Use PowerSCADA Expert One-Lines	118
One-Line Flowchart	118
What Are PowerSCADA Expert One-Lines?	120
Where are the files located?	120
One-Line Device (zOL)	120
One-Line Colors	121
Assigning Colors	121
Add INI Settings to AdvOneLine.ini and Citect.ini	121
AdvOneLine.ini Settings	121
Citect.ini Settings	123
Start and Stop One-Lines	123
Work with Genies	123
Genie Type Descriptions	124
Configure an Automatic Transfer Switch (ATS)	124
Transfer Switch Information: Left/Right/Bottom	124
Display Information	124
Configure a Busbar	125
Configure a Circuit Breaker or Switch	125
Configure a Meter	126
Configure a Source	126
Configure a Transformer	127
GenieConfiguration.xml File	127

SupportedGenies.xml File	127
Work in the Configuration Utility	128
One-Line Configuration Utility	128
Modify AdvOneLine.csv	128
Review Genie Configurations	128
One-Line Errors and Warnings	130
Communication Errors	130
Error Logging	130
Perform Repairs for One-Lines	131
Automatic Transfer Switch (ATS) Information	132
ATS Information	132
ATS Errors	133
Busbar Information	133
Busbar Errors	134
Breaker and Switch Information	134
Breaker and Switch Errors	134
Meter Information	135
Meter Errors and Warnings	135
Source Information	136
Source Errors and Warnings	136
Transformer Information	137
Transformer Errors	137
Work with Alarms	138
Alarms Overview	138
PC-Based Alarms	138
Onboard Alarms	138
Set Up Alarm Pages	139
Add Setpoints and Delays	139
Set Up an Alarm Based on an Enumeration	139
Change an Alarm Severity	140
Enable Waveforms for Onboard Alarms	140
Set Parameters for Event Log Length and Historical Logging of Events	140
Event Storage: [Alarm]SummaryLength Parameter	140
Event Logging: [PLSEventLog] Parameter	141
Add an Onboard Alarm Tag	141
Set Up Audible Alarms	142
Restore and Back Up Projects	142
Restore a Project	142
PowerSCADA Expert Backup	143
Use Cicode to Customize a Project	143
Use Cicode in PLS_Include	143
PLSProviderEngine.ci Module	143
Module Construction	143
CallProvider	143
GetProviderStatus	144
GetProviderResult	145
Clear Cache and Refresh Platform	145
PLS_ClearCache	145
PLS_PlatformRefresh	146
Customize Default Behaviours	146
Time Zone Settings	146
Time Synchronization	147
Trend Tag Scan Intervals	147
Disk Storage Calculation for Trends	148
Deadbands and Ignored Devices and Topics	148

Waveform Management	150
Waveform Storage	150
Waveform Database and Special Waveform Tags	150
The Runtime Environment	152
Launch and Close the Runtime Environment	153
View the Graphics Page in the Runtime Environment	153
Log On to the Runtime Environment	153
View the Interface	153
Lockout/Tagout Icon	154
Communications Loss	155
View the Alarms/Events Page	155
Equipment Column	155
Filter Information	156
Remove/Insert/Move Columns	156
Sort by Column	156
The Event Log	156
The Alarm Log	156
Unacknowledged Alarms and Disabled Alarms	157
Alarm and Events Logging	157
Acknowledge, Silence and Print	157
When Alarms do not Display Correctly	158
Event/Alarm Log Columns Table	159
Alarm/Event Filter Form	160
Use Security Viewer	161
Security Viewer Filter	162
Use the Analysis Page	163
Use the Equipment Pop-Up Page	164
Perform IEC61850 Advanced Control	165
View Waveforms	165
Enter Setpoints for Alarms	165
View Real-Time Trends	166
View Lists of Real-Time Information for the Genie	166
Override Tag Status	166
Perform IEC 61850 Advanced Control	167
EcoStruxure WebServices (EWS)	169
Use Basic Reports	169
Set up the Runtime Environment for Basic Reports	170
Create the Menu Items for Report Page	170
Add the Following INI Parameters	170
Set Up a Display Client for Basic Report Viewing	171
Tasks in the Basic Reporting Application	171
Create and View Basic Reports	171
Run a New Basic Report	171
Run a Basic Report and Save its Configuration	172
View a Basic Report Using a Saved Configuration	172
Modify and View a Basic Report Using a Saved Configuration	173
Remove a Saved Configuration	173
Configure Email Settings to Send Basic Reports	173
Send Basic Reports via Email	174
Scheduling Basic Reports	176
Read, Export, Print, and Edit Basic Reports	176
Toolbar Options	177
Export a Basic Report	177
Edit the Basic Report Appearance	178
PowerSCADA Expert Basic Reports	178
Single Device Usage Reports	178
Multi Device Usage Reports	179

Tabular Reports	180
Trend Reports	180
Enable Windows Authentication for Basic Reporting	181
URL Routing for Basic Reports	182
Windows 2008 R2 and Windows 7	182
Windows XP	182
Localization for Basic Reports	182
Template Editor	183
Use LiveView	183
LiveView Tables	184
LiveView Basic Readings Summary	184
LiveView Power Flow Summary	184
LiveView Energy Summary	185
LiveView Energy Readings	185
LiveView Fundamental Phasor Readings	185
LiveView THD Current Summary	185
LiveView THD Voltage Summary	186
LiveView Uptime Summary	186
LiveView Incremental Reactive Energy Summary	186
LiveView Incremental Real Energy Summary	187
LiveView Harmonic Apparent Power Flows	187
LiveView Harmonic Reactive Power Flows	188
LiveView Harmonic Real Power Flows	188
LiveView Demand Current Summary	189
Live View Demand Voltage Summary	189
LiveView Viewer	189
Open LiveView from a URL Link	190
LiveView Viewer Display	190
Where's My Device?	191
Missing Topics	191
Clear Cache and Platform Refresh	192
Set Up LiveView	192
Create Menu Item for LiveView Page	193
Create a LiveView Template	194
LiveView Formatting	194
LiveView Placeholders	195
LiveView Formulas	196
LiveView Thresholds	197
Modify LiveView Template	198
Duplicate LiveView Template	198
LiveView Delete	199
LiveView Template Localization	199
Enable Windows Authentication for LiveView	199
Rapid Access Labels	200
Before You Begin	200
Create the Sticker	201
Read the Sticker	201
Troubleshooting	202
Multi-Monitor Support	202
The Demo Mode	202
Application Configuration Utility	203
Application Services	204
Diagnostics	204
EcoStruxure Web Services Setup	204
Avoiding EWS Provider Timeouts	205
Event Notification Service	205
Application Services Host—Citect Data Platform	205

Applications	206
One-Line Engine	206
Basic Reports	207
Single Sign On	208
Single Sign-On (SSO) Calls from a Web Client	208
Assign and Control User Privileges	209
Default User Access Settings (Privileges)	209
Add and Modify User Accounts	210
Use Windows Integrated Users	210
Change Access Rights	211
Appendix 1: Default Genie Library	212
PLS_ALARM	212
PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2	212
PLS_ANSI_CB_1 / PLS_ANSI_CB_2	214
PLS_ANSI_SW_1 / PLS_ANSI_SW_2	215
PLS_DISPLAY	215
PLS_GEN_BASE_1 / PLS_GEN_BASE_2	216
PLS_GEN_CMD_1 / PLS_GEN_CMD_2	217
PLS_GEN_EQ_1 / PLS_GEN_EQ_2	217
PLS_IEC_BASE_1 / PLS_IEC_BASE_2	218
PLS_IEC_CB_1 / PLS_IEC_CB_2	218
PLS_IEC_SW_1 / PLS_IEC_SW_2	219
PLS_METER	219
ITEM1	220
Appendix 2: Citect INI Parameters	221
Parameters Database	221
General PowerSCADA Expert Parameters	222
watchtime	222
kernelStatisticUpdateRate	222
UseWriteMultiRegistersOnly	222
timeout	223
retry	223
RetryTimeout	223
RetryException	224
standbyRefreshRate	224
standbyCheckTime	224
statusUnitCheckTime	225
initUnitCheckTime	225
initCacheTimeout	225
cacheRefreshTime	226
TimeSync	226
StatusRegister	226
StatusRegistersCount	227
StatusRegisterType	227
ModbusBase	227
RegMode	228
timeZone	228
Alarm Parameters	229
UsePLSFilter	229
Performance Tuning Parameters	229
Bandwidth Allocation Parameters	229
BandwidthAllocation	230
Packet Blocking Optimisation Parameters	230
enableScatteredReads	230
percentBlockFill	230
maxBlockSize	231
minBlockSize	231

Tag Scan Rate Parameters	232
HighScanRate	232
LowScanRate	232
Advanced Tag Block Capabilities (Invalid Memory Access Blocks defined)	232
Invalid Block Tag Definition Syntax	233
Configuration Notes	234
Write-only Memory	235
Tag Blocking Notes	236
Write-only Tags	236
Waveform Parameters	237
WaveformsDisable	237
WaveformMatchMargin	238
WaveformCheckTime (PM, CM, and Sepam)	238
WaveformZone (Sepam)	238
Sepam Event Reading Parameters	238
EventTable	238
EventIdle	239
MicroLogic Modules Configuration Parameters	239
CCM	239
Module-Specific Packet Blocking Optimisation Settings	239
MicrologicV INI Settings	240
Data Replication Parameters	240
Database root folder path	240
Database root UNC path	240
Replication destination configuration	240
Graphics Library Parameters	241
Maximum number of entries that can be held in Event Log	241
Parameters for Alarm and Event States	241

Integration Parameters	242
Appendix 3: Additional INI Parameters	244
Quantum PLC Time-Stamped Events	244
Appendix 4: Glossary	245
Appendix 5: Frequently Asked Questions (FAQs)	254
If I don't use PowerLogic drivers, how do I create device profiles?	254
How should we manage categories and subcategories?	254
When should I create a device type rather than device profile?	254
How do we synchronize a new PC with the master Profile Editor PC?	254
What do I do before I add or remove devices in the Profile Wizard?	254
What are the requirements for device names?	255
How do I troubleshoot device communications issues?	255
How do I use MODBUS communications methods?	256
How can I add more than one device at a time?	256
What are the naming conventions for servers and clusters?	256
How and when do I create users for the Runtime environment?	256
How do I manage projects in the Citect Explorer environment of PowerSCADA Expert?	257
On the Graphics page, what do I need to know about creating genies?	258
How do we customize existing templates?	258
How do I change the default pickup/dropout text for alarms?	259
What can I modify during runtime?	260
Why do the browser navigation buttons not work?	260
What can I set up in logging and archiving?	260
How do I create and configure busbars?	260
What INI parameters should I use for debugging?	261
How do I tune my system for best performance?	262
If a tag is configured, how is it polled in the device?	263
Device popup from a one-line: Why do the fields overlap?	264
Can I change the %CLUSTER% name in the Profile Wizard?	264
When viewing PowerSCADA Expert from a web client, I see a licensing error. How do I fix it?	264
A device can prevent writes to its registers: how do I ensure that writes are successful?	265
How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes?	265
How do I create an audit in the Event Log for user logins and logouts?	265
Why do I get an "out of memory" message when I try to run the Automation Interface?	266
Why am I seeing #COM for circuit breaker status in the genie status page?	266
Why can't I acquire waveforms in the waveform viewer?	266
Why won't the Excel DBF Add-In toolbar install?	267
What causes the "First dbf record" error message? How do I keep it from happening?	267
Why is my device in comms loss?	267
How do I set up select before operate?	267
Index of Terms	268

Introduction

Welcome to the help file for PowerSCADA Expert, v8.0. To navigate through the file, you can use the search options on the left. Additionally, you can access other related PowerSCADA Expert help files.

In this section, you will find these topics:

[PowerSCADA Expert—A Complete Solution on page 1](#)

[Assumptions on page 1](#)

[What is SCADA? on page 1](#)

[What is PowerSCADA Expert? on page 2](#)

[Prepare for PowerSCADA Expert on page 2](#)

[Components of a Project on page 2](#)

[Configuration Tools on page 2](#)

[Runtime Environment on page 3](#)

[Supported Device Types and Protocols on page 3](#)

[System Requirements and Installation Instructions on page 5](#)

[Uninstall and Reinstall PowerSCADA Expert on page 5](#)

[Quick Start: Typical Workflow on page 5](#)

[IEC 61850 System Setup Workflow on page 8](#)

[Get More Information on page 9](#)

[Manuals on page 9](#)

[Help files on page 9](#)

[Customer support on page 9](#)

PowerSCADA Expert—A Complete Solution

The PowerSCADA Expert system provides a complete solution for the monitoring and control of any electrical distribution network. Using PowerSCADA Expert 8.0 as the engine, this product 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 on page 1](#)

[What is SCADA? on page 1](#)

[What is PowerSCADA Expert? on page 2](#)

Assumptions

The person who installs the product will have received training in the entire PowerSCADA Expert 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 PowerSCADA Expert?

This product includes several tools that use the PowerSCADA Expert 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 PowerSCADA Expert system.
- In the One-Line Configuration utility, you can review genie configurations, and then make necessary repairs before you compile your project.
- 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.

Prepare for PowerSCADA Expert

For a full discussion about preparation for a project, see **Plan a Project** in the CitectSCADA.chm help file (Program Files > Schneider Electric > PowerSCADA Expert > v8.0 > bin). This section of the help discusses physical layout, requirements such as architecture and security, and project design.

Components of a Project

For detailed information about the components that make up a project, see **Components of a project** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin). 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 project. This tool is simply a means of moving device profile information into the project and converting it into formats that PowerSCADA Expert 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 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.

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 the product, 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 PowerSCADA Expert 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. For JBus devices, select Generic JBus Device.
- Sepam 20, 40, and 80 Range, 2000
- Masterpact MicroLogic 5P and 6P, A, H
- Compact NSX (MicrologicV)
- CM2000
- CM4000 series
- PM650
- PM800 series
- PM5000 series
- PM700 series
- ION protocol devices
- IEC 61850 protocol devices
- IEC 870-5-104
- DNP3
- BCPMA (branch circuit power meter, full feature support)
- CSI SER (Cyber Sciences SER)

How Do Drivers Work?

The following paragraphs describe driver subscriptions. For each unique tag request made, the I/O server adds one point to the point count. Tag subscriptions are limited based on the point count in the license. Exceeding the subscribed point count will ultimately cause the I/O server to shut down.

Two Subscription Types

There are two different types, one used between the graphics level and I/O server, and one for polling devices and cache refreshing. The subscription between drivers and polling devices does not increase point count. Only the subscription that begins at a client system and ends up in the I/O server will increase point count. Via this

subscription, requests are sent to the drivers with value changes propagating all the way back to the client system. The client system could be the display client, alarm server, trend server and so on. What a driver then chooses to do with the requests—in terms of coupling this to a physical request to a field device—can differ, depending on the protocol. Some simple protocols propagate the request straight through to the field device; others have their own polling scheme to the field device and merely service the driver requests from a cache.

Subscription Expirations

If a tag is no longer being read, the cache refreshes in this manner: Graphics client subscriptions are immediately unsubscribed when the graphics page is closed. Although most drivers release subscriptions if no client is requesting them, the IOServer is capable of background polling (configurable on a per-device basis). These tag subscriptions are not released, and the driver still polls them. However, they are not counted anywhere, because nothing is consuming the data for those tags on the IOServer. On the other hand, once a subscription goes against the point count, it remains in the count as long as the project is running.

Expiration is immediate if no clients are subscribed to the tag. An "expiration time-out value" is not configurable.

Edit Driver Parameters

Certain IEC 61850 devices may have driver parameters associated with them. You can edit the datasets and report control blocks that will then be exported to PowerSCADA Expert.

To begin editing driver parameters: from the Create Device Profiles tab, click the Parameters sub-tab.

To begin editing datasets, click Edit in the DataSets line. Follow instructions in [Edit IEC 61850 Datasets on page 44](#) for help.

To begin editing report control blocks, click Edit in the Report Control Blocks line. Follow instructions in [Edit IEC 61850 Report Control Blocks on page 45](#) for help.

Problems with Duplicate Devices using PowerLogic Drivers

If you have duplicate devices that use PowerLogic drivers, you will have system problems. The PowerLogic drivers that are affected are:

- Generic Power Device
- MicroLogic
- MicroLogic A
- CM4000
- PM800
- SEPAM20
- SEPAM40
- SEPAM80

A duplicate device is created when two or more I/O devices in a system communicate with a single physical device. For this to be true, the I/O devices would have the same IP address, same TCP port, and same MODBUS device address.

System performance would be affected, and there would be problems with onboard alarms. More seriously, it could result in a system crash.

System Requirements and Installation Instructions

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

Uninstall and Reinstall PowerSCADA Expert

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

- PowerSCADA Expert 8.0 (if you uninstall this, you also uninstall the Profile Editor)
- PowerSCADA Expert Profile Editor
- Any additional PowerSCADA Expert programs, such as the WebServer, that you installed

If you uninstall programs 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\PowerSCADA Expert 8.0. The first time you launch the application after you re-install it, it will locate the project data and re-link it.

Quick Start: Typical Workflow

The following is a brief description of the steps you will take when you create a (Modbus or non-61850) system. Also, see [Typical Workflow Illustration on page 13](#) for a flow chart that illustrates the Profile Editor workflow.

NOTE: If you are upgrading from a previous version, you should run the Migration Utility. See [Use the Migration Utility on page 10](#).

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 the product. For individual tag conversions, you can also add "exceptions" (mostly used in WAGES topics).
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 that have already been created, 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 within PowerSCADA Expert, and to move them to the location where they can be imported into PowerSCADA Expert.
12. **Use the Profile Wizard to set up clusters and add devices to PowerSCADA Expert:** Use the Profile Wizard to set up clusters (created in step 9). See [Add a Device to a Project on page 92](#) for additional information.

NOTE: To use the one-line graphics, you must add a memory device called zOL to the project. See [One-Line Device \(zOL\) on page 120](#).

Use the Profile Wizard to add the exported devices, one at a time; or use the Automation Interface ([Add Multiple Devices through the Automation Interface on page 102](#)) to add all of the devices from one page (example found in: Program Files\Schneider Electric\PowerSCADA Expert 8.0\Bin\Example Automation.xls). You will need device networking information for this step.

13. **Compile and correct errors in PowerSCADA Expert:** 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 controls the 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. **To configure :** Click Tools > One-Line Configuration to launch. Use the configuration utility to validate the project one-line, and to configure the project for animation.

NOTE: Each time you make a change to the one-line, after the system is set up, you need to run the configuration utility again to update the one-line. See [One-Line Flowchart on page 118](#).

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



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



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

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



Ensure that a startup function is defined. You must call PLS_StartAdvOneLine () on all machines where the zOL device is running. (See [One-Line Device \(zOL\) on page 120](#) for information about the zOL device).

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.

NOTE: If you choose Multi-Process (Server and Control Client), you must also choose **Networked (connect to other SCADA computers)** at the following screen. Otherwise, reporting functionality may be affected.

20. **Set up reporting, Live View tables, and Rapid Access Labels:** Add any reports and report configurations that you want (see [Use Basic Reports on page 169](#)). Add any Live View views that you want (see [Use LiveView on page 183](#)). If desired, set up rapid access labels (see [Rapid Access Labels on page 200](#)).
21. **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. Ensure that all commands are tested and verified.

WARNING

UNINTENDED EQUIPMENT OPERATION

- Ensure that all commands are being correctly interpreted by each device.
- Verify that the device is physically locked out/tagged out before performing work on it or any downstream equipment.
- Also consider the possibility of communications loss that could yield false readings.
- Ensure that all safety regulations and procedures have been followed before you work on the equipment.

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

IEC 61850 System Setup Workflow

These are the basic steps you need to follow to set up an IEC 61850 device in your 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 48](#)).
2. Import the first ICD file into the Profile Editor (see [Import Filter Screen on page 59](#)).
 - a. Create the device type.
 - b. Match or verify tags for PowerSCADA Expert.
 - c. Complete the import.
3. Create a device profile for the IEC 61850 device type (see [Add an IEC 61850 Device to a Project on page 98](#)).
 - a. If needed, add/edit datasets and report control blocks (see [Edit IEC 61850 Datasets on page 44](#) and [Edit IEC 61850 Report Control Blocks on page 45](#)).
 - b. Select the appropriate tags for PowerSCADA Expert 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 PowerSCADA Expert, and to SCL.
 - PowerSCADA Expert 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 project.
 - a. From within PowerSCADA Expert, add a new project.
 - b. Add the appropriate clusters, networks, and servers.
9. Using the Profile Wizard or Automation Interface, add your devices to PowerSCADA Expert.

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

For more information, see [Add an IEC 61850 Device to a Project on page 98](#).
10. Compile and run the project.

Get More Information

For additional information, use one of these options:

[Manuals on page 9](#)

[Help files on page 9](#)

[Customer support on page 9](#)

Manuals

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

- **Vijeo Citect Installation 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 this product, there are several related help files. They are located in the PowerSCADA Expert Bin folder: \Schneider Electric\PowerSCADA Expert 8.0\Bin).

Customer support

The following is the contact information for customer support:

Schneider Electric

35, rue Joseph Monier

CS 30323

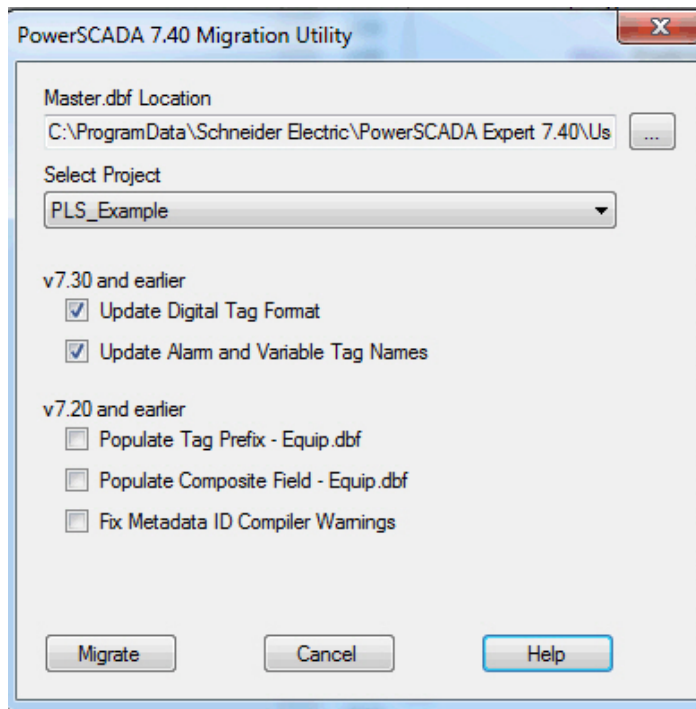
F - 92506 Rueil Malmaison Cedex

www.schneider-electric.com

Use the Migration Utility

The migration utility allows you to migrate previous versions of PowerSCADA Expert to the current version. You only need to run this utility one time. The table below describes the changes that will be made.

1. Before you run the migration utility, back up your system.
2. Locate the .exe file for the utility. It is on:
InstallDisk:\Setup\SetupSupport\Extras\Upgrade\.
3. Run the file named *PSEMigrationUtility*.



- a. In the Select Project drop down list, choose the project you want to migrate to the current version.
- b. Check the boxes for the elements you want to update (see table below).
- c. Click Migrate.

When migration is complete, a summary screen displays. It lists the results of the migration, including updates and errors.

Element	Description	Changes
v7.30 and earlier:		
Update Digital Tag Format	This option will update all DIGITAL tags in Variable.dbf to FORMAT "##."	This causes digital tags in PowerSCADA Expert to display without decimals ("1" or "0," but not "1.000."
Update Alarm and Variable Tag Names	Previously, some device-specific tags were renamed to fit a generic naming convention for version 7.40. This option renames all existing tags to the new convention names.	Check this box to rename all new convention names. For example, the old "Sepam Not Reset" is now "Generic Not Reset."

Element	Description	Changes
v7.20 and earlier:		
Populate Tag Prefix-Equip.dbf	Equipment Name, which was used to build tag names, is now the equipment hierarchy name (can no longer be used to build tag names)	TagPrefix field added. It is now used to build tags. If the TagPrefix field is empty, IODevice name is used to populate Tag Prefix. If IODevice name is also empty (in a composite device), EquipmentName is used IF there are no periods in the name.
Populate Composite Field - Equip.dbf	The Parent field (previously used to determine the parent piece of equipment) has been removed from the .dbf file.	The Composite field replaces the Parent field. The Composite field will display the Parent field information, if applicable.
Fix Metadata ID Compiler Warnings	The Cicode function StrToLocal no longer allows partially translated text. For example, in <i>@(Protection), 2</i> , "Protection" must be translated. Also, "2" is the metadata ID; in all custom fields (1-8) of all alarm tags, the ID part of the field must be removed.	All custom fields in alarm tags will remove the ID part (1-8) of the field, IF the translation identifier is present. Thus, in <i>@(Protection), 2</i> the "2" is removed; it will be changed to <i>@(Protection)</i> .

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:

[Overview of the Profile Editor on page 13](#)

[Typical Workflow Illustration on page 13](#)

[Launch the Profile Editor on page 17](#)

[Add Engineering Unit Templates, Units, and Conversions on page 19](#)

[Define Device Types and Tags on page 23](#)

[Device Type Screens and Workflow on page 23](#)

[Print the .CSV File on page 29](#)

[Add Custom Tags and Tag Addresses on page 31](#)

[Add, Edit, or Delete a Device Type on page 26](#)

[Edit Tag Addresses on page 34](#)

[Edit Generic Tag Addresses on page 39](#)

[Edit Functional Addresses on page 31](#)

[Create Device Profiles on page 39](#)

[Enable Waveforms on page 39](#)

[View Device Profiles on page 39](#)

[Add Edit or Delete Device Profile on page 41](#)

[Edit IEC 61850 Datasets on page 44](#)

[Edit IEC 61850 Report Control Blocks on page 45](#)

[IEC 61850 System Setup Workflow on page 8](#)

[Create Data Concentrator Device](#)

[G3200 Device Setup on page 48](#)

[Set Up Projects in the Profile Editor on page 50](#)

[Project Screens and Workflow on page 50](#)

[The Set Up Projects Tab on page 51](#)

[Add, Edit, or Delete a Project on page 52](#)

[Create a Composite Device Type](#)

[Edit and Delete Information in a Project on page 54](#)

[Customize Tag Names on page 54](#)

[Import and Export Project Files on page 54](#)

[Tag Types on page 65](#)

[Customize Tag Names on page 54](#)

[IEC 61850 Tag Construction on page 65](#)

[Define an Enumeration on page 66](#)

[Format Code Definitions on page 67](#)

[Logic Codes on page 69](#)

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 [Define Device Types and Tags on page 23](#) 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 [Create Device Profiles on page 39](#) 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 [Set Up Projects in the Profile Editor on page 50](#) for complete instructions.

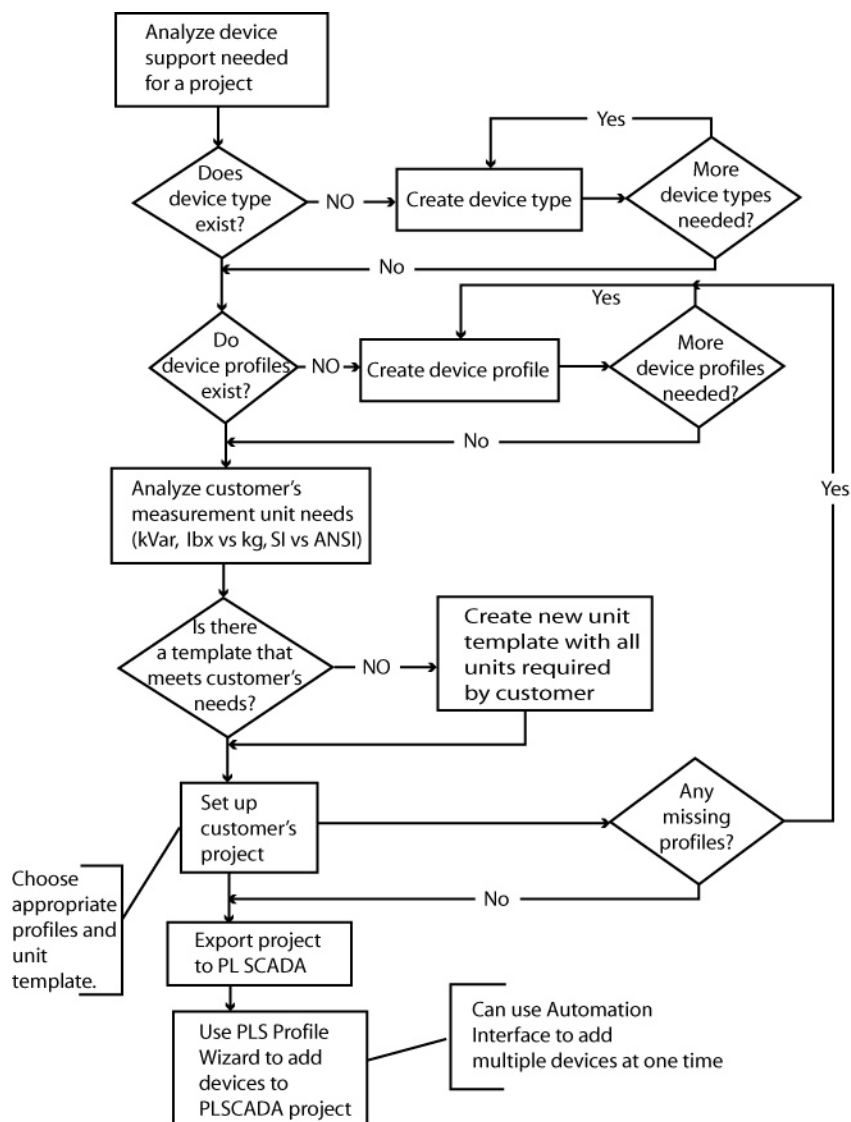
This product uses the IEC61850 tag-naming convention to create tags that measure device quantities. Although most of the tags you will use are already entered into the system, you can add custom tags. For more information, see [Tag Types on page 65](#).

Typical Workflow Illustration

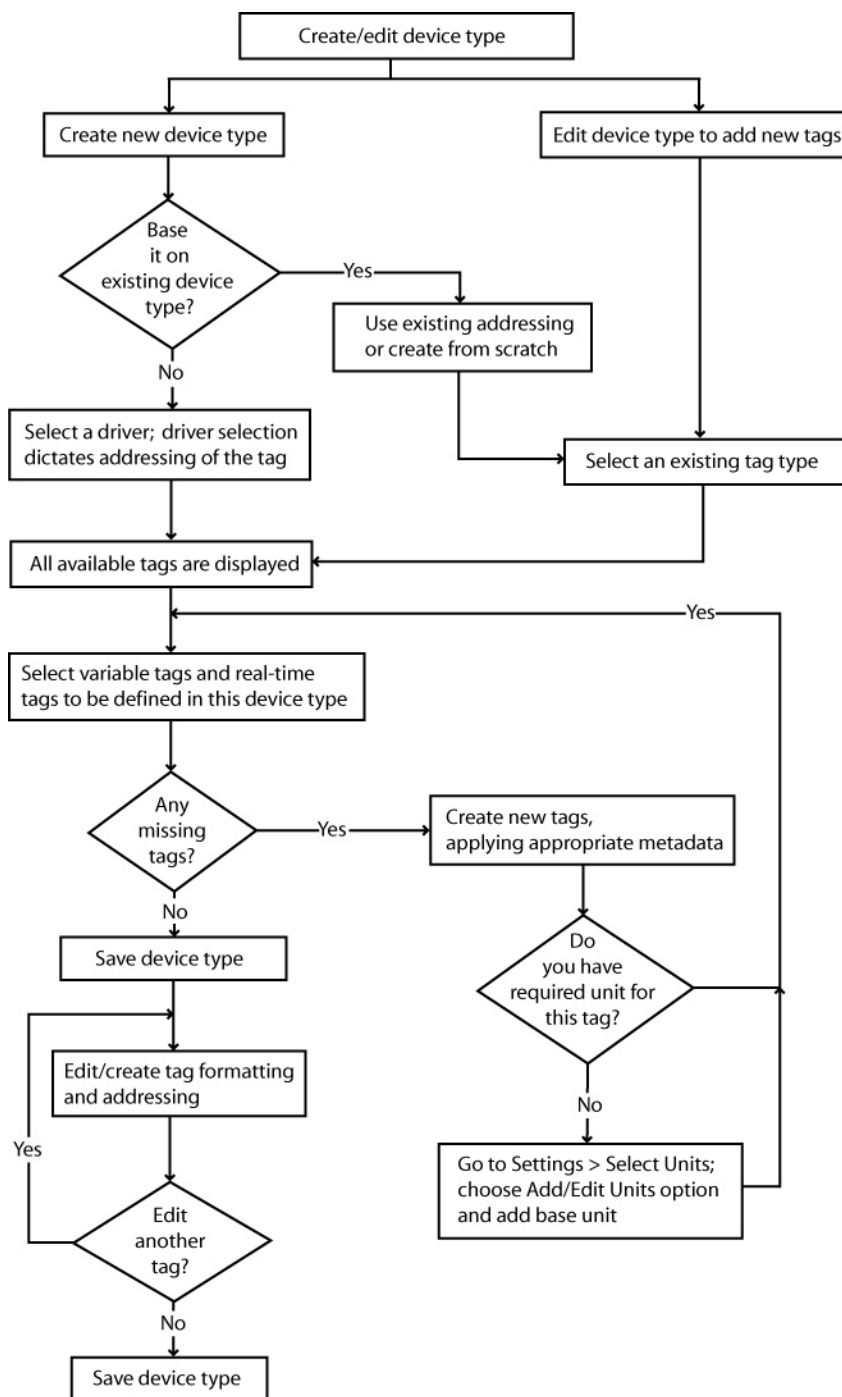
The following four flow charts illustrate the use of the Profile Editor. The first illustration provides an overview, and the following illustrations show:

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

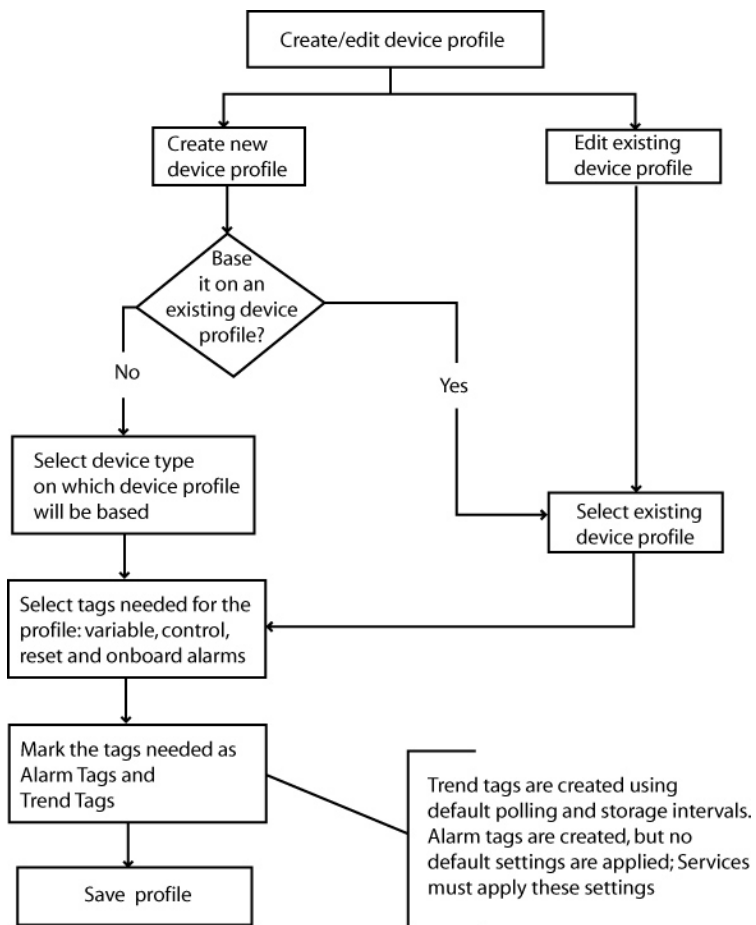
Workflow Overview



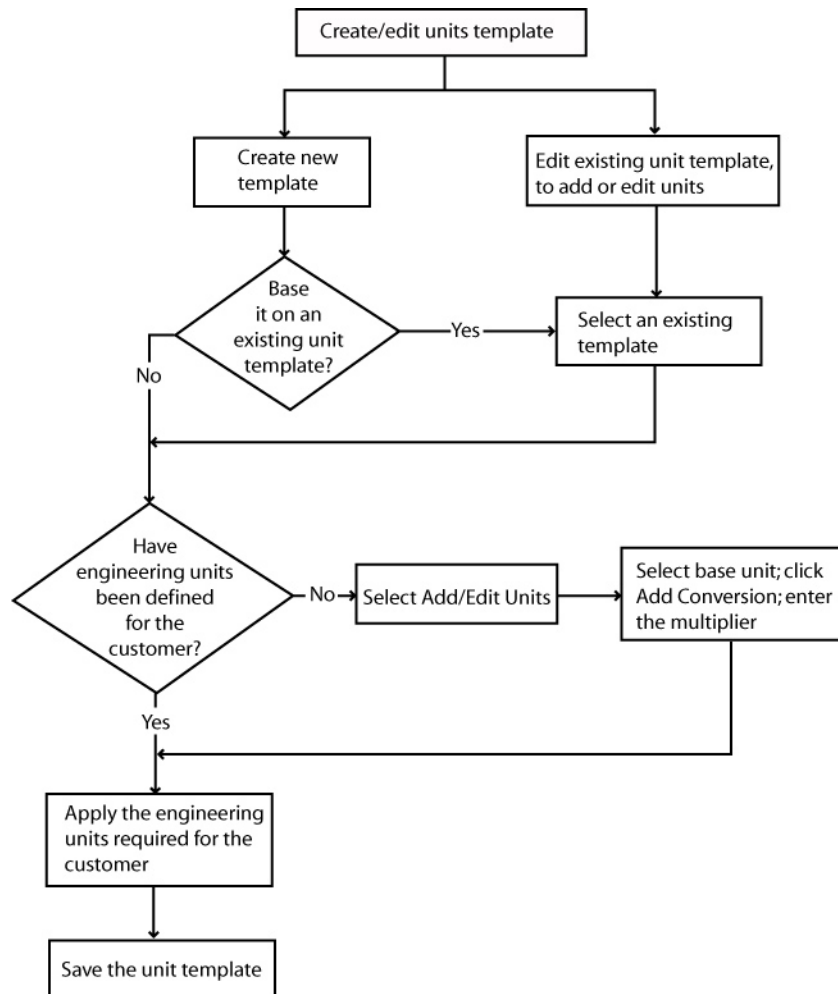
Create/Edit Device Type



Create/Edit Device Profile



Create/Edit Unit Templates



Launch the Profile Editor

To launch the Profile Editor, click Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 > 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.

Set 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.

Profile Editor 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. To begin importing, click File > Import. For more information, see Import and Export Project Files on page 54.

Field Name/Valid Entries	Comments
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. To begin exporting, click File > Import. For more information, see Import and Export Project Files on page 54
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 64 .
Settings > Set Up Custom Tags	Displays the Add/Edit Custom Tags screen. See Set Up Custom Tags on page 31 for a description of this screen.
Settings > Set Up Device Type Categories	Displays the Set Up Device Type Categories. See Set Up Device Type Categories on page 29 for a description of this screen.
Settings > Set Up Engineering Unit Templates	Displays the Set Up Engineering Unit Templates screen. Click Set Up Engineering Templates and Select Conversions on page 19 for a description of this screen.
Settings > Set Up Trend Definitions	Displays the Set Up Trend Definitions screen. Click Set Up Trend Intervals on page 43 for more information.

Add 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:

[Set Up Engineering Templates and Select Conversions on page 19](#)

[Add or Edit a Base Engineering Unit or Conversion on page 22](#)

Set Up Engineering Templates and Select Conversions

Use the Set Up Engineering Unit Templates screen when you want to add, edit, or delete an engineering unit template, or to make changes to how the unit is reported.

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). When you have finished making change, click Save & Exit.

Field Name	Valid Entries	Comments
Template 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. Click to edit an engineering unit or its conversion.
	Delete	Only available if you have added a template. 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 <i>Create From</i> 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.
Unit Template Name	This field is blank if you selected <i>Create New</i> or <i>Create From</i> ; type the name of the new template. A name displays if you have selected a template to edit; you can change the name. A name displays, but it is greyed out if you selected a template to delete. Click <i>Save</i> to save the changes you make.	When creating a new template or creating from an existing template, type the name of the new template. To change the name of an existing template, choose it from the <i>Unit Template to Delete</i> 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.
Default Units Subtab		
Use this subtab to manage unit templates and to add global changes to a unit.		
Base 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 Add or Edit a Base Engineering Unit or Conversion on page 22 .
Abbreviation	n/a	Added for the unit when the selected unit was created. To edit a user-created unit, see Add or Edit a Base Engineering Unit or Conversion on page 22 .
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 Add or Edit a Base Engineering Unit or Conversion on page 22 . 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.

Field Name	Valid Entries	Comments
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 Add or Edit a Base Engineering Unit or Conversion on page 22 .
Offset	n/a	Used for units that have more than one scale. For example, for temperature, if the base is degree Celsius, and you want to offset to Fahrenheit, you would type 32 here (and 1.8 in the multiplier).
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.
Unit Exceptions Subtab		
Use this tab to apply "exceptions" for individual tags, changing the way the unit is reported for the tag(s). This is most commonly used for WAGES tags.		
Tags	Choose an individual tag or tag subgroup.	This tag will be reported with the new settings.
Options	1. From the dropdown list, choose the unit you want to use for this tag/tag group. 2. Click the radio button for the exception to be made. 3. Either double-click the tag, or click the right arrow to move it to the Exception list.	1. If you choose Apply Unit Conversion, the tag will be reported according to the unit you select. For example, if you want to report Air Volume in gallons, rather than cubic meters, choose "gallon" from the Select Unit dropdown list. 2. Click "Apply Unit Conversion" to convert and report the tag according to the unit you selected. Click "Apply Unit Name Only" to add the unit name to it, but not convert it, when it is reported.
Exception List	Review your changes.	You can check or uncheck tags here, changing them from one conversion option to the other. When you uncheck a tag, you do not remove it, you change it from being converted to simply being reported according to the unit you selected.

Apply Conversions

Use this screen to apply unit conversions to a template. To add a new conversion, see [Add or Edit a Base Engineering Unit or Conversion on page 22](#).

To apply 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. In the Selected Unit column, click the down arrow and select the conversion you want to use. Fahrenheit to Celsius temperature conversions are handled by offsets (see [Add or Edit a Base Engineering Unit or Conversion on page 22](#)).
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.

Delete 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.

Add or Edit 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).

Click Settings > Set Up Engineering Unit Templates. At the Set Up Engineering Units screen, choose the template you want to edit, and click Add/Edit Units. The following table describes the fields of the Add/Edit Units screen. Instructions for editing and deleting units are after the table.

Field Name	Valid Entries	Comments
Unit Conversions	n/a for pre-defined units/conversions (greyed out) 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.
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 live when you select a custom unlocked unit.
Base Unit	When editing a unit/conversion, select the unit from this drop-down menu. When adding a new base unit, type the name.	Used in the Profile Editor only; not passed to projects for graphics viewing.
Base Unit Abbreviation	Type the abbreviation for the selected base unit.	If there is no conversion, this is passed to projects for viewing graphics.
Conversion Unit	Type the name of the conversion unit, such as milliamps, when amps is the base unit.	Becomes live only when you highlight a unit. Used in the Profile Editor only; not passed to projects for graphics viewing.
Conversion Abbreviation	Type the abbreviation for the conversion unit.	This is passed to projects for viewing graphics.
Multiplier	Use this field to determine the number of base units that are in the conversion unit. 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, $17.3 \times 1,000 = 17300$ bytes

Field Name	Valid Entries	Comments
Offset	Use this field to determine a numeric offset.	Example: If degrees Celsius is the base unit, and you are creating a conversion unit for Fahrenheit, you would enter a multiplier of 1.8 and an offset of 32.

Edit 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.

Delete a Base Engineering Unit or Conversion

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

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.

Define 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 page 23](#)

[Use the Define Device Type Tags Tab on page 24](#)

[Add, Edit, or Delete a Device Type on page 26](#)

[Print the .CSV File on page 29](#)

[Set Up Device Type Categories on page 29](#)

[Edit Functional Addresses on page 31](#)

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 [Add or Edit a Base Engineering Unit or Conversion on page 22](#).
2. To manage the tags you will use, see [Define Device Types and Tags on page 23](#).

3. To add and edit custom tags, see [Add Custom Tags and Tag Addresses on page 31](#).
4. To add or edit device types, see [Add, Edit, or Delete a Device Type on page 26](#).
5. To establish device type categories and subcategories, used in reporting, see [Print the .CSV File on page 29](#).
6. To edit tag addresses, see [Edit Tag Addresses on page 34](#).

The PowerSCADA Expert system uses the IEC61850 tag naming convention. For information about the types of IEC61850 tags and their configuration, see [Tag Types on page 65](#).

Use 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. 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:  	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 17 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.) 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. 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.

Field Name/Valid Entries	Comments
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 Tag Types on page 65 for more information.
Type (Real Time only)/Display only	Displays the data type chosen when the tag was created.
Address (not Control tags)/ 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. Tag Types on page 65 for a detailed description of address construction.
Normally Closed (Control tags only)/ 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.) 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.) 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). See Edit Tag Addresses on page 34 . 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 settings, you are prompted to go to the Edit Address screen.
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 Edit Tag Addresses on page 34 .
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. Note: Functional addressing is described in Appendix 4: Glossary on page 245 .
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. A simple example: Address = "T:MV;m:" + (startingpoint + 1005).ToString() + ";L:P:22"

Field Name/Valid Entries	Comments
	You would then define "startingpoint" when instantiating 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) Utility Type (real-time only) Statistical Type (real-time only) Quantity (real-time only)	Each of these types is a real-time filter, added when the tag was created. See Set Up Custom Tags on page 31 for more information.
Categorization (onboard alarm only) Subcategorization (onboard alarm only) Alarm Type (onboard alarm only) Alarm Group (onboard alarm only) Alarm Level (onboard alarm only)	Each of these is an onboard alarm folder, added when the tag was created. See Set Up Custom Tags on page 31 for more information.

Add, Edit, or Delete 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 the Define Device Type Tags tab; then 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.

For instructions on editing or deleting device types, see the steps below the table.

Field Name	Valid Entries	Comments
Create New	Click one of the radio buttons to select the action you want to take.	Click to add a device type that is not based on an existing type.
Create From		Click to copy an existing device type.
Edit Existing		Click to edit an unlocked device type.
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 when you choose Create From. Check this box if 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: maximum 32 characters, do not use \ / : * ? < >	If creating a device type, type the name. If editing a device type, the device type that was selected for editing displays here. You can change the name here.
Lock this Device Type	Check to lock device, so that it cannot be edited.	This action cannot be undone. You cannot edit a locked device type. If it is a standard device type, you cannot edit or delete it. If you added the device type, you can delete it but not edit it.

Field Name	Valid Entries	Comments
Device Category	Choose the category for this device.	To create categories, see Print the .CSV File on page 29 . In addition to predefined categories, you can add custom categories. See Set Up Device Type Categories on page 29 for instructions. Categories are used in the Device Creation wizard, and are a means of shortening the list of devices you must view.
Subcategory	Choose the subcategory for this device, if needed.	As with categories, subcategories are created in the Print the .CSV File screen.
Driver	Select the driver for the device type.	Predefined drivers are 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. Use the Generic Power Device driver for third-party devices. The Controllable Device driver is currently not used. Use Generic JBus Device driver for JBus devices.
Display Associated Profiles	(Active only in Edit mode) Click to display a list of profiles that are associated with the selected device type.	This list is used 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 click this button. Note the profile(s) associated with the device type. 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 (such as 100ms, Onboard Alarm, Power Factors).
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). 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 Add Custom Tags and Tag Addresses on page 31 for instructions. 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.

Edit 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 [Edit Functional Addresses on page 31](#)).

Real Time Tags sub-tab: you can edit the address (see [Edit Tag Addresses on page 34](#)) and choose a different format.

- 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.

Delete 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.

Assign 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:

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),

Print 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.

Set 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.

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.

Field Name	Valid Entries	Comments
	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. If editing or deleting, select the name from the drop-down menu. 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. If editing or deleting, select the name from the drop-down menu. You cannot delete a subcategory that is associated with a device type. Predefined subcategories do not display. Currently, the predefined subcategories are: Protection Device, Monitoring Device, and PLC.

Add 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.

Edit 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.

Delete 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.

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

Edit Functional Addresses

Use this feature to add variables to addressing. You can re-use a variable 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.

Add Custom Tags and Tag Addresses

PowerSCADA Expert 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:

[Set Up Custom Tags](#)

[Edit Tag Addresses on page 34](#)

[Edit Tag Addresses](#)

Set 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. The Add/Edit Custom Tags screen displays. The following table describes the fields.

Instructions for editing or deleting a custom tag 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 Tag to Edit 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. See Delete a Custom Tag on page 34 for instructions on using this button.
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 or 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 (such as 100ms, controls, currents) plus onboard alarms, resets, and controls.
Data Type	Select the data type.	These are PowerSCADA Expert tag data types. They affect the logic codes that are available for display in the Edit Address screen. See Logic Codes on page 69 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 for display purposes in PowerSCADA Expert graphics pages. It determines where the decimal displays.

Field Name	Valid Entries	Comments
		Choose the reporting format, to be used in PowerSCADA Expert, from ## to #0.#####. 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 PowerSCADA Expert uses when reading data from devices. 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.		
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 (dropdown lists are 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.
Alarm Filters tab (dropdown lists are 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.

Edit 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.

Delete 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:

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.

Edit 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 [Tag Types on page 65](#) and [Logic Codes on page 69](#).

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 69](#). 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.

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 Logic Code:	You can edit this field either here or in the Add/Edit Tag screen.

Field Name	Entry	Comments
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 69 .
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). 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.
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. The value will be scaled in this manner: Value x 10x where X = the scale. 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). Conversion factors take this form: #####E##. For example, 123E-2 becomes 123x10-2 which becomes 1.23.
Offset	$y = ,x + b$	y = the final value reported by PLSCADA b = the offset m = the conversion factor x = the original value in the meter b = rarely used, mainly in temperature conversion 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. 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.

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, PM5000, 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, PM800, and PM5000 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.

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.

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 69 .

Box Name	Field Name	Comments
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. 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	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:

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 69 .
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. The value will be scaled in this manner: Value x 10x where X = the scale. 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. 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	n/a	Not used for digital controls.

Edit 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.

Add 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.

Edit Generic Tag Addresses

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

The variable tag properties used in this screen are described in a topic in the Citect help file. For detailed information, see **Variable Tag Properties** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

Create 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 [Define Device Types and Tags on page 23](#)). 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:

[Enable Waveforms on page 39](#)

[View Device Profiles on page 39](#)

[Add Edit or Delete Device Profile on page 41](#)

[Select Trend Intervals on page 43](#)

[Set Up Trend Intervals on page 43](#)

[Edit IEC 61850 Datasets on page 44](#)

[Edit IEC 61850 Report Control Blocks on page 45](#)

[IEC 61850 System Setup Workflow on page 8](#)

[Create a Composite Device Type on page 46](#)

[Create Data Concentrator Device on page 48](#)

[G3200 Device Setup on page 48](#)

[DNP3 Protocol Support on page 50](#)

Enable 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 CET manual. For PowerLogic devices, reference the PMCU help file.

As device information is polled and received by PowerSCADA Expert, the waveform becomes available for viewing. See *The Alarm Log* in [View the Alarms/Events Page on page 155](#) for information on viewing waveforms in the runtime environment.

View 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 [Enable Waveforms on page 39](#) 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.
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.	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)	n/a	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)	n/a	
Alarm Type (PC Based and Onboard Alarm)	n/a	
Alarm Group (PC Based and Onboard Alarm)	n/a	
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 (such as real-time, trend) to view the tag groups included in this device profile.

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

Add Edit or Delete 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.

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.
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 [Add Project Parameters on page 54](#).
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 (such as description, metadata).
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 PowerSCADA Expert 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. Click Finish.

When the project is added to the project, PC based alarms are added to the Analog Alarms or Digital Alarms file. When the project is added to the project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the 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 147](#).

To change a trend interval for a tag, see [Select Trend Intervals on page 43](#) To add additional trend tags, see [Set Up Trend Intervals on page 43](#).

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 [Edit IEC 61850 Datasets on page 44](#) and [Edit IEC 61850 Report Control Blocks on page 45](#) 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.

Edit a Profile

Only unlocked profiles are available for editing.

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.

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 [Set Up Trend Intervals on page 43](#)).
 - PC Based Alarms or Onboard Alarms sub-tabs: change alarm levels (this will override the default that is set in [Set Up Custom Tags on page 31](#)).
 - Onboard Alarms sub-tab: enable waveform capture for on-board alarms (see [Enable Waveforms for Onboard Alarms on page 140](#) 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 [Set Up Custom Tags on page 31](#) for more information).
 - Parameters sub-tab: Edit parameters for IEC 61850 driver parameters (see [Edit Driver Parameters on page 4](#) 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.

Delete a 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.

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. Exit the screen.

Select 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 [Set Up Trend Intervals on page 43](#).

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.

Set 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. For detailed information, see **Trend Tag Properties** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).
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.

Edit 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.

Create 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.
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.

Create 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.

Copy 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.

Edit and Delete 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.

Edit 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.

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.

Create 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 PowerSCADA Expert).
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.

Create 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.

Copy 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.

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

Edit and Delete 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.

Create 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 PowerSCADA Expert combines the functionality of the multiple devices, end users only need to consider a single device when analysing a location in their system.

Cyber Sciences Sequence of Events Recorders (SER)

For instructions on setting up and using Cyber Sciences SERs, refer to the system technical note (STN) entitled *How can I Use Cyber Sciences SERs with PowerSCADA Expert?*

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, and so on. 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) 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.

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 PowerSCADA Expert 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.

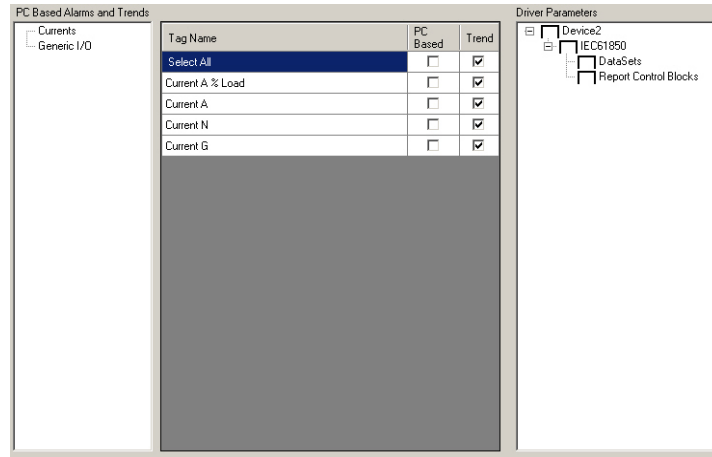
When the profile is added to the project, PC based alarms are added to the Analog Alarms or Digital Alarms file.

When the profile is added to the project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the 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 147](#).
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 [Edit IEC 61850 Datasets on page 44](#) and [Edit IEC 61850 Report Control Blocks on page 45](#) for information on editing these two 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.

Create Data Concentrator Device

When you create a data concentrator and/or a data-concentrated devices in the Profile Editor, all of the related devices must use the same protocol. Examples of data concentrators are PLCs that use inputs from various devices or an RTU that concentrates data from multiple devices.

To add a data concentrator to your project, follow these steps in the Profile Editor:

1. From the Define Device Type Tags tab, add a custom device type for the data concentrator. Use the Generic Power Device driver.
2. Add the tags that are specific to the data concentrator (such as device date and time).
3. Add addresses for any custom tags you created.
4. Add the data-concentrated device. Use the Generic Power Device driver, as you did for the data concentrator.
5. Add the tags for the data-concentrated device (such as currents, voltages, and breaker status).
6. Add addresses for these tags (or add functional addressing for them).
7. Repeat steps 4 through 6 for additional data-concentrated devices.
8. From the Create Device Profiles tab, add a device profile for each data-concentrated device type you included.
9. From the Set Up Projects tab, add the profiles to a project.

G3200 Device Setup

Use these instructions to set up a G3200 gateway in PowerSCADA Expert.

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, and so on).
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 PowerSCADA Expert and SCL exports.

In CET850

Create the CID file for the G3200 gateway.

In PowerSCADA Expert

1. Open the Profile Wizard.
2. To the System Devices, add an IEC 61850 data concentrator for the G3200.
 - a. Enter instance information screen, select the BRCBs that you need.
 - b. Select the CID file you created in CET850.
 - 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.

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 CET850

Create the CID file for the G3200 gateway.

In PowerSCADA Expert

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 CET850.
 - d. Complete the remaining steps in the Profile Wizard.

DNP3 Protocol Support

You can create device types and profiles that use the DNP3 protocol. See [Add a DNP3_TCP Device on page 96](#) or [Add a Serial Device to a Project on page 94](#) 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.

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).

Set 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 PowerSCADA Expert 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 on page 50](#)

[The Set Up Projects Tab on page 51](#)

[Add, Edit, or Delete a Project on page 52](#)

[Create a Composite Device Type on page 46](#)

[Edit and Delete Information in a Project on page 54](#)

[Customize Tag Names on page 54](#)

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 PowerSCADA Expert is installed and the corresponding PowerSCADA Expert project has been created, this also copies the file that is used by the Device Creation Wizard to the PowerSCADA Expert 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 in all device profiles in the project for which you have created the customized tag. For this change to be in the PowerSCADA Expert project: you need to delete the device profile from that 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, add 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 PowerSCADA Expert project: you need to remove the device profile from that 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, which you can add via 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 [Customize Tag Names on page 54](#).
- **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 [Add Project Parameters on page 54](#).

To add or edit project information, click Add/Edit. The Add/Edit Project screen displays. See [Add, Edit, or Delete a Project on page 52](#) 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 [Export a Project on page 55](#) 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:



Add, Edit, or Delete 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 added to the project. From this screen you also export individual projects to the format that can be added to PowerSCADA Expert (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 PowerSCADA Expert, click the Display Projects button:



A list displays with the projects that have been added (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 [Set Up Engineering Templates and Select Conversions on page 19](#) 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 [Add or Edit a Base Engineering Unit or Conversion on page 22](#) 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 245](#).)
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.
10. Give the device type a Configured Name. This name might indicate its status (which driver it uses, whether it has a functional address, and so on) 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.

Edit 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.

Delete 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.

Edit and Delete Information in a Project

After you have exported a project to a PowerSCADA Expert project, you still need to use the Device Creation Wizard to add system information to the PowerSCADA Expert project. See [Before You Use the Profile Wizard on page 87](#) for information about this process.

Customize 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.

Add Project Parameters

The Project Parameters sub-tab allows you to add optional lines of information about this project. This information can be used for verification or identification when you want to import the project.

To add project parameters:

1. From the Set Up Projects tab, click the Project Parameters sub-tab.
2. On the first available line, type a name and value for this information. Example: If you want to track versions, in the Name field, you might type "Version." Then, in the Value field, type the appropriate version for this project.

The new parameter is added. It will help you identify the project when you want to import it into another instance of the Profile Editor.

Import and Export 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 project; see [Export a Project on page 55](#).
- 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 57](#).
- 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 56](#).
- Import a project from another instance of the Profile Editor or from an IEC61850 file.
- Import SCL files. You can 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 PowerSCADA Expert profile.
- Import ICD files. You can import either functional constraints or report control blocks.

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 [Import Files into the Profile Editor on page 57](#).

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 64](#).

Export a Project

Use this feature to export a project from the Profile Editor into PowerSCADA Expert.

If the Profile Editor is on the same computer with PowerSCADA Expert, and if you have created a matching project in the PowerSCADA Expert project, this process will copy all project data—device tags, device types, and device profiles—into that project. If the Profile Editor is not on a computer with PowerSCADA Expert, you need to manually move the exported file to the PowerSCADA Expert server. See **To Move 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 PowerSCADA Expert project, you need to create a matching project in PowerSCADA Expert. To do this:

1. Launch the instance of PowerSCADA Expert to which you will be exporting data: Click Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 > PowerSCADA Expert 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 PowerSCADA Expert (to ensure that it correctly exports to its matching project).

To export a Profile Editor project to the PowerSCADA Expert 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 PowerSCADA Expert option. (The selected export(s) are indicated beneath the Export Project button.)
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\v8.0\Projects.
- Each Profile Wizard profile file is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v8.0\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\v8.0\WizardProfiles\[project name]\Profile Wizard.

In PowerSCADA Expert, 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.

To Move Files if the Profile Editor is not with the Server (Reuse of a Project)

When the Profile Editor is not on the same computer as the PowerSCADA Expert 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 PowerSCADA Expert server computer, paste Equipment.profiles in this location:

[Drive Letter]:\Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.0\User\[Project]

where:

[Drive Letter]: is the drive on which you installed the PowerSCADA Expert server 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 PowerSCADA Expert (see **Before you export**, above).

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

Profile Editor Export

To back up a project file, see [PowerSCADA Expert Backup on page 143](#).

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 or later 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 [Customize Tag Names on page 54](#) and [Add Project Parameters on page 54](#) 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:\Program Data\Schneider Electric\PowerSCADA Expert\v8.0\Applications\Profile Editor\Projects\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.

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 that the export did not succeed is listed for the device types shown.

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

[Project Drive]\Program Data\
SchneiderElectric\PowerSCADA Expert\v8.0\Applications\Profile
Editor\WizardProfiles\<project name>\SCL Export\sclFileName.iid).

Import 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 Profile Editor projects, you can import tags, device types, and profiles. 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 other file (SCL, CID) that you want to import.
3. From the Profile Editor, click File > Import.
4. At the Import File Selection window, click browse (), 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 64](#).) 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.
6. When you locate the desired file, click Open to choose it. Then click Start Import.

The system analyses the import and attempts to match imported items with existing items on the local machine.

If you are importing a .pls file, skip to step 8.

If you are importing IEC 61850 data, the Import Filter screen displays. Use this screen to perform an initial filter on functional constraints or on report control blocks. See [Import Reconciliation Screen on page 60](#) for more information.

7. Make your selections, then click Continue.
8. When the Import Reconciliation screen displays, you can begin the process of matching or rejecting individual tags. See [Import Reconciliation Screen on page 60](#) for a description of the parts of this screen.
9. On the Import Reconciliation screen, click an item in middle pane. Respond to the item according to your preference for it. You must set the status first for units, then tags, and finally the device type.
10. After you match or ignore 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 64](#) for instructions on creating, using, and deleting import templates.

Import 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 need to reconcile mismatches; and data will be available for creating 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).
- **PowerSCADA Expert profile:** The data will then follow the normal rules for the profiles in this project.

Import Filter Screen

This screen displays after you choose an IEC 61850 file to import (.ICD, .CID, or .IID extension) and click Start Import. Use this screen to begin filtering data for import. You choose whether to filter on functional constraints or report control blocks.

Functional Constraints

1. Click the Functional Constraint button.
2. Choose the functional constraints that you want to include.

The filters the list of devices for which you will import data to those that contain one or more of the selected functional constraints.
3. Check the device(s) that you want to include.

Report Control Blocks

1. Click the Report Control Block button.

The list of devices and their related report control blocks that are included in the import file displays in the middle column.
2. Check the devices and/or related report control blocks that you want to include in the import. If you check a device, all of the report control blocks under it are included.

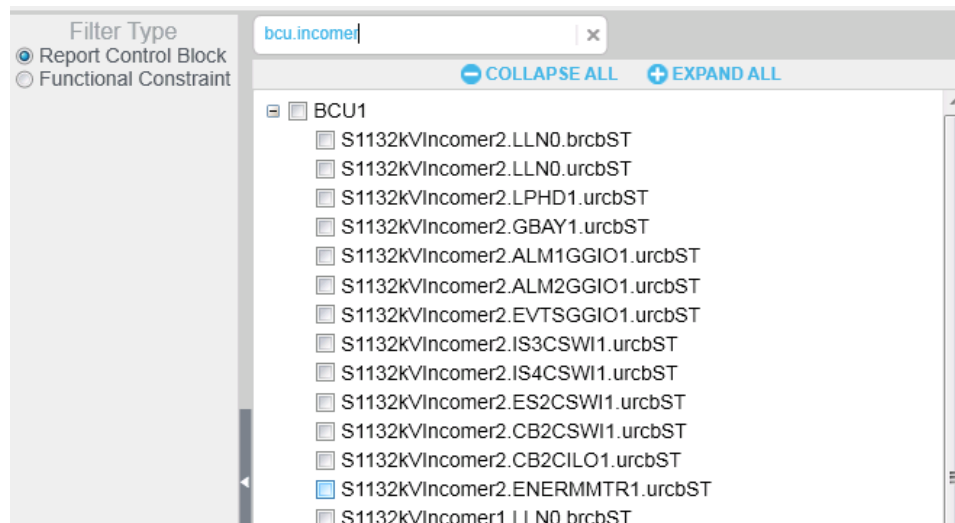
The right-hand column displays the IEDs/report control blocks that you have selected.

NOTE: Use the filter above the middle pane to search. You can enter partial names separated by dots to further shorten the list.

The figure below illustrates an example in which a search was done first on "bcu" and then on "incomer" (note that entries are not case sensitive). The search string would be:

bcu.incomer

The result would look something like this:

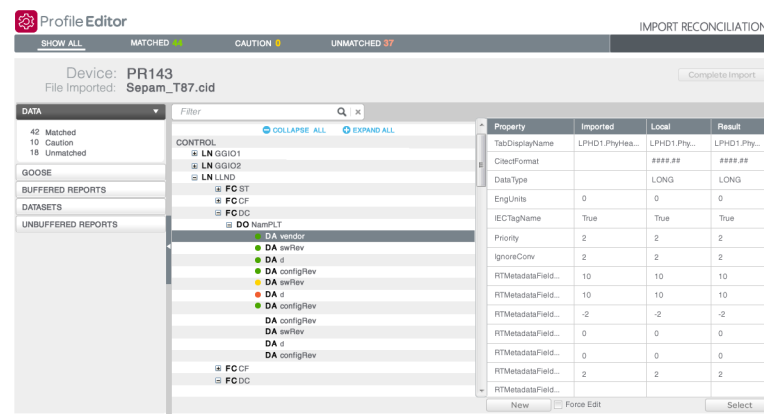
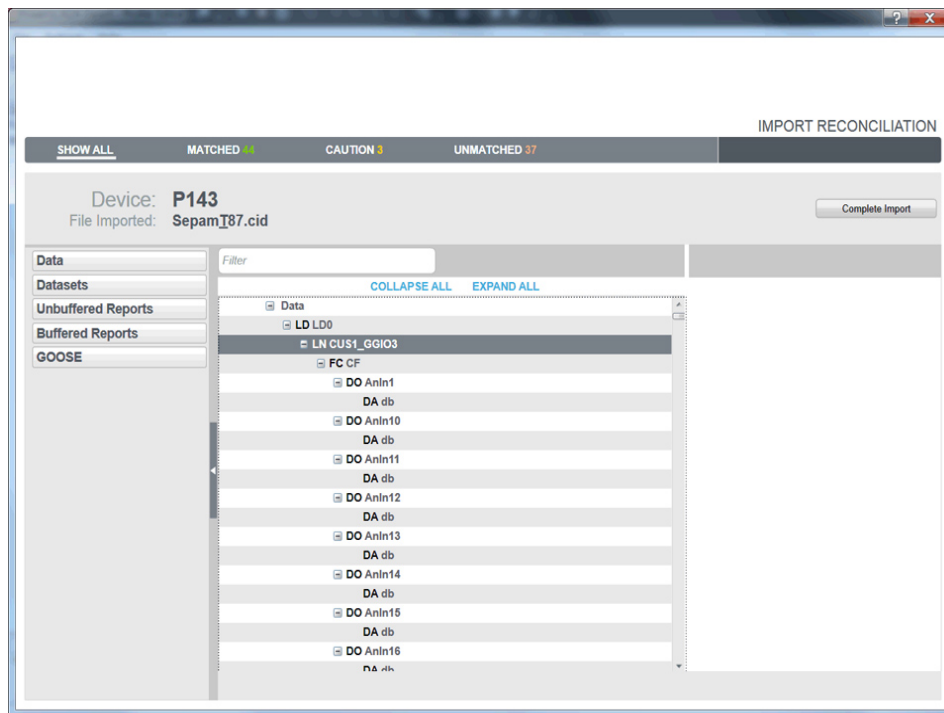


When you have selected either the functional constraints or report control blocks, click Continue. The data is filtered on the last filter option that you chose (you cannot combine filters).

The Import Reconciliation screen displays. See [Import Reconciliation Screen on page 60](#) for help finishing the import.

Import Reconciliation Screen

Use the Reconcile Import Screen to find matches for the items you are importing and to filter import tags to determine whether items are matched or not matched. The first figure below shows the screen before import is complete. The second one shows the results after import has been completed.



The screen is divided into three panes:

Left-hand pane:

The selections made in this pane provide an initial filter for what you view in the middle pane (see below). The tree view at the top shows the imported file data categories:

For .pls files imported from the Profile Editor, the categories are: device profiles, device types, tags, and units.

For IEC61850 files, the categories are: Data, Datasets, Unbuffered Reports, Buffered Reports, and GOOSE.

Select a category to filter the list in the middle pane to only the items belonging to that category.

To further filter the middle pane, click one of the matched status lines (matched, partially matched, unmatched) to view only items of that status. The number of items in that status also displays.

Middle pane:

This pane shows a tree view with data.

Filter: To filter on a specific item, type the name (such as phsA for phase A current). The entry can be the exact name, or you can enter a partial name or even a wildcard (*). The filter is not case sensitive.

The data in the middle pane is filtered to include only the items for the tag you specify. To clear the filter so you can enter a new one, click the "x" beside the filter box.

Collapse All/Expand All: Click *Collapse All* to collapse all nodes on the screen. Only the top-level nodes will display. Conversely, click *Expand All* to open all nodes, displaying all of the information on all nodes.

The bottom section of the middle pane displays, in tree form, the data that you selected in the left-hand pane:

- For files imported from the Profile Editor (.pls files), you can view: Show All, Device Profiles, Device Types, Tags, and Units.
- For IEC 61850 files, you can view: Data, Datasets, Unbuffered Reports, Buffered Reports, and GOOSE.

The bullets indicate:



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



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



: no match; item does not match any local item

Items that have no icon beside them are ignored during the import.

Re-match Items within a Logical Node

Because IEC 61850 tags are often imported with prepended information (logical node: LN) that prevents the import from matching them, you may find several unmatched items. You can use the re-match feature to enable matching for them.

To do this, right-click the logical node where the unmatched items are found, and choose Re-match.

The import feature will then exclude the logical node, and use the remaining information in the item name to find matches. In the screen shown above, it would include functional constraint (FC) **ST**, data objects (DO) **PhyHealth** and **Proxy**, and five data attributes below them:

ST.PhyHealth.q

ST.PhyHealth.t

ST.Proxy.q

ST.Proxy.stVal

ST.Proxy.t

Right-hand pane:

This pane illustrates the status of each of the tags. Click a tag and read the information for it:

- **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; by default, this item is inherited from the local status.

New: At the bottom of the list, click this button to add an item as a custom topic. The Add/Edit Custom Tag screen displays for you to create the tag.

Force Edit: Check Force Edit to display a screen that lets you edit the item's information. You can 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.

Select: After importing, you can manually match an unmatched item. To do this, highlight the tag in the middle pane, the type matching information in the Search field in the upper right corner of the screen. Choose the matching item and click Select. This yields an unverified match (yellow bullet). To confirm the match, click Match on the right.

Complete the Import

1. Right click the first item that you want to review/change in the middle pane, then select the option for how you want the import to handle this item. Each item's status controls the options you will see:

Item Status	Right-Click Options	Description
Ignored	New	The custom tag screen opens for you to add the attributes for a new tag.
Unmatched	Ignore, New	Ignore: Changes the status so that the import will exclude this tag. New: The custom tag screen opens for you to add the attributes for a new tag.
Matched	Ignore, Set to Unmatched	Ignore: The import will exclude this tag. Set to Unmatched: The tag is no longer matched; but the import will not succeed. All unmatched tags must be matched or ignored before you complete the import.
Partial Match	Ignore, Set to Unmatched, Match	Ignore: The import will exclude this tag. Set to Unmatched: The tag is no longer matched; but the import will not succeed. All unmatched tags must be matched or ignored before you complete the import. Match: The tag attributes will change to the information you see in the Results column.

2. Continue through all of the items until you have set the match status for each one.
3. Click *Complete Import*.

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.

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.

Tag Types

This product 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:

[IEC 61850 Tag Construction on page 65](#)

[Define an Enumeration on page 66](#)

[Format Code Definitions on page 67](#)

[Logic Codes on page 69](#)

[Block Writes on page 85](#)

IEC 61850 Tag Construction

IEC 61850 tags are flexible, which allows them to specify how functions are implemented in devices. The IEC 61850 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 IEC 61850. For these devices, the general IEC61850 formatting was followed when creating tags.

If you are writing Cicode (see [Use Cicode to Customize a Project on page 143](#)). You will need to know the IEC 61850 tag name that you added to the device profile for that device. You can print the .csv file to view tag names (see [Print the .CSV File on page 29](#)). Apart from that, you would only need to add tags if you are installing a third-party device that is not standard to PowerSCADA Expert. 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)
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 IEC 61850 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.

Define 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

Bit x Bit y	Status	Circuit Breaker Position	Returned Value
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).

Use 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:

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 [Edit Tag Addresses on page 34](#) 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 212](#)), or create a custom genie (see [Create New Genies on page 116](#)).

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 69](#) for templates of constructed tags.

Real-Time Format Code Definitions

Attributes	Modifiers	Comments
T (type) Required	SS = single status DS = double status enumeration ST = string UT = UTC time MV = measured value (float) CM = complex measured value (float) BC = binary counter (integer)	Temporarily, this may return a string; when PowerSCADA Expert is upgraded to handle large integers, this will change.
D (module— Micrologic devices)	B = BCM P = PM M = MM	

Attributes	Modifiers	Comments
	C = CCM	
M/m/S/s/C/c/I/i (register type)	M = holding registers in hexadecimal m = holding registers in decimal S = input coil (status register) in hexadecimal s = input coil (status register) in decimal C = output coil (writable only) in hexadecimal c = output coil (writable only) in decimal I = input register (read only) in hexadecimal i = input register (read only) in decimal	
Register Number Modifiers (register number from 1–4)	u## = ## registers are unsigned; ## is a decimal 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: RegisterValue x scale For SS and DS: there must be a 1U default; the modifier will be a bitmask: - The mask must use hex only, 16 bits/register - Attach the ones, then the zero mask, to the register; if you only have ones masks, just attach them - 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 (1 = high, 2 = normal, 3 = low)	Defines the priority PowerSCADA Expert uses in processing data.
V (conversion factor)	Use scientific notation without the decimal.	Examples: 354E-3 = 0.354 354E1 = 3540 Will be multiplied before the value is returned.
L:P (logic code) 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 as L:I for ION. For logic code descriptions, see Logic Codes on page 69

Alarm Format Code Definitions

Attributes	Modifiers	Comments
T (type) Required	ALM = alarm	

Attributes	Modifiers	Comments
D (module—optional for Micrologic devices)	B = BCM P = PM M = MM C = CCM	BCM is straight addressing, and therefore, optional.
F (file) Required	File number will be in decimal, up to 5 digits	
Q (unique ID) 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) 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) Required	NO = normally open NC = normally closed 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.

Custom Control and Reset Format Codes

Attributes	Modifiers	Comments
C (command) Required	NO = normally open NC = normal closed 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 69. Write Address format: T:SS;m:##;L:P:101 Example: C:NO;T:SS;m:1234;1;L:P101;T:SS;m:3456;1;L:P101		

Logic Codes

Logic codes tell the PowerSCADA Expert 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.

Logic Code	IEC Type	PowerSCADA Expert 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: High byte = Month 1–12 Low byte = Day 1–31 Register N+1: High byte = Year 0–199 (+1900) Low byte = Hour 0–23 Register N+2: High byte = minutes 0–59 Low byte = seconds 0–59
Date / Time (L:P:2) (6 register)	UT	LONG	6 sequential registers	No	Generic - if it fits	Register N: Seconds 0–59 Register N+1: Minutes 0–59 Register N+2: Hours 0–23 Register N+3: Day 1–31 Register N+4: Month 1–12 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: High byte = Month 1–12, Low byte = Day 1–31 Register N+1: High byte = Year 0–199 (+1900) Low byte = Hour 0–23 Register N+2: High byte = minutes 0–59 Low byte = seconds 0–59 Register N+3: msec = 0–999 (unused)

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
Date / Time (L:P:4) (3 or 4 registers SEPAM)	UT	LONG	3 or 4 sequential registers	No	SEPAM	Register N: Bits 0–6 = Year: 0–70 (2000–2070) 71–99 (1971–1999) Register N+1: Bits 8–11 = Month Bits 0–4 = Day Register N+2: Bits 8–12 = Hour Bits 0–5 = Minutes Register N+3: msec = 0–59,999 (seconds are ms/1000)
Date/Time (L:P:5) 3-register Micrologic	UT	LONG	3 sequential registers	No	micro	Register N: High byte = Month 1–12, Low byte = Day 1–31 Register N+1: High byte = Year 0–69 (+2000), Year 70–99 (+1900) Low byte = Hour 0–23 Register N+2: High byte = minutes 0–59 Low byte = seconds 0–59
Date/Time (L:P:6) 4-register Micrologic	UT	LONG	4 sequential registers	No	micro	Register N: High byte = Month 1–12, Low byte = Day 1–31 Register N+1: High byte = Year 0–69 (+2000), Year 70–99 (+1900) Low byte = Hour 0–23 Register N+2: High byte = minutes 0–59 Low byte = seconds 0–59 Register N+3: msec = 0–999 (unused)
Date/Time (L:P:7) 3-register Argos	UT	LONG	3 sequential registers	No	Argos	The number of seconds since 01/01/2000 (00:00:00) register 1 = MSB

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						register 2 = LSB register 3 = milliseconds
Date/Time (L:P:8) 4-register IEC 870-5-4	UT	LONG	4 sequential registers	No	generic	Register N: Bits 0–6 = Year: 0 – 127 (2000– 2127) Register N+1: Bits 8-11 = Month Bits 0-4 = Day Register N+2: Bits 8-12 = Hour Bits 0-5 = Minutes Register N+3: msec = 0-59,999 (seconds are ms/1000)
Modulo 10k (L:P:10)	BC	STRING	Up to 4 registers	No	generic	Result is a string representation. Range is 0 to 9,999,999,999,999,999 9 Each register has a range of 0 to 9,999 Result is: $-R4 \cdot 10,000^3 + R3 \cdot 10,000^2 + R2 \cdot 10,000 + R1$
Modulo 10k Val (L:P:11)	BC	REAL	Up to 4 registers	No	generic	Result is a string representation. Range is 0 to 9,999,999,999,999,999 9 Each register has a range of 0 to 9,999 Result is: $-R4 \cdot 10,000^3 + R3 \cdot 10,000^2 + R2 \cdot 10,000 + R1$ 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 (L:P:12)	BC	STRING	Up to 4 registers	No	generic	Result is a string representation. Range is 0 to 9,999,999,999,999.9 Each register has a range of 0 to 9,999

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						Result is $-(R4 \cdot 10,000^3 + R3 \cdot 10,000^2 + R2 \cdot 10,000 + R1) / 1000$
Modulo 10k Energy Val (L:P:13)	BC	REAL	Up to 4 registers	No	generic	Result is a string representation. Range is 0 to 9,999,999,999,999.9 Each register has a range of 0 to 9,999 Result is $-(R4 \cdot 10,000^3 + R3 \cdot 10,000^2 + R2 \cdot 10,000 + R1) / 1000$ 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 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. Second register is masked to test for either one 1 or one 0. Result is: 0 = off and 1 = on. 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: Result is 0 = intermediate, 1 = off, 2 = on, 3 = bad-state. 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: Result is: 0 = false and 1 = true. 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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						Second register is masked to test for either one 1 or one 0. Result is: 0 = off and 1 = on. 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: Result is: 0 = intermediate, 1 = off, 2 = on, 3 = bad-state. 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: Result is: 0 = false and 1 = true. 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.) Result is: 0 = off and 1 = on. 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.) 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: Result is: 0 = intermediate, 1 = off, 2 = on, 3 = bad-state.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						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: 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: Result is: 0 = false and 1 = true. 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: Result is: 0 = false and 1 = true.
Status Int (L:P:29)	BC	LONG	1 register	No	CM/PM	One register is bitanded with one mask. The result will be an integer that can be used to choose the appropriate enumeration.
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.) Result is a combination of the results for each register, using this formula: 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/C M	REAL	3 registers	No	CM/PM	First register (300–399 inclusive) indicates that this is a analog input register. Second register is treated as a signed value.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						Third register can contain a value from – 3 to 3 and will be used to scale the second register ($R2 \cdot 10^R3$). 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.
Breaker Rack Status (L:P:230)	EN	LONG	2 to 3 registers	No	Generic	Register 1 = breaker racked in Register 2 = breaker racked out Register 3 = breaker in test (optional) Results: 0 = racked in 1 = racked out 2 = test 3 = error 4 = in between positions
PL Analog Output (L:P:31)	MV/C M	REAL	2 registers	No	CM/PM	First register (400–499 inclusive) indicates that this is an analog output register. Second register is treated as a signed value. 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/C M	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)

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						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 –2,147,483,648 to –2,147,483,647. Values can be scaled using a fixed scale or a scale register. 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/C M	REAL	1 to 4 registers	Either (optional)	Generic	For a single register: treated as an unsigned value from 0 to 65,535. 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 0 to 4,294,967,295. Values can be scaled using a fixed scale or a scale register. 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/C M	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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						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/C M	REAL	2 registers	Either (optional)	Generic	Same as Scaled Register except that 0xFFFFFFFF or 0x00007FFF will be NA. 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 B (L:P:36)	MV/C M	REAL	2 registers	Either (optional)	Generic	Same as Scaled Register except that 0xFFFFFFFF will be NA. 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/C M	REAL	2 sequential registers	No	Generic	Uses the IEEE standard for floating-point arithmetic (IEEE 754); register 1 is MSB, register 2 is LSB 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/C M	REAL	1 register	Either (optional)	Generic	For a single register: treated as a signed value from -32,767 to +32,767: From the value of the unsigned register, subtract 32768; then apply the scale.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						0000 or FFFF will be NA. 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/C M	REAL	1 to 4 registers	Either (required)	Generic	Result is: $R1 + \dots + Rn * 10^{\text{scale}}$ 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/C M	REAL	3 registers	Either (required)	Generic	Result is: $R1/R2 * R3 * 10^{\text{scale}}$ If R2 is zero, result will be #COM 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/C M	REAL	1 to 4 registers	Either (required)	Generic	Result is: $R1 * \dots * Rn * 10^{\text{scale}}$ 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/C M	REAL	1 to 4 registers	Either (required)	Generic	Result is: $\text{Avg}(R1 \dots Rn) * 10^{\text{scale}}$

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						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/C M	REAL	2 to 4 registers	Either (required)	Generic	Result is: $\text{Avg}(R1 \dots Rn-1) * Rn * 10^{\text{scale}}$ NOTE: This logic code (as with 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/C M	REAL	2 registers	Either (required)	CM/PM	Result is: $(R1 * 10^{\text{scale}}) + R2$ NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Sum with Scale Unsigned (L:P:46)	MV/C M	REAL	2 registers	Either (required)	CM/PM	Result is same as above, except unsigned. NOTE: This logic code (as with 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/C M	REAL	2 registers	Either (required)	Generic	Result is: $\text{sqrt}(R1^2 + R2^2) * \text{scale}$ NOTE: This logic code (as with 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/C M	REAL	4 registers	Either (required)	Generic	Result is: $\text{sqrt}([R1 R2]^2 + [R3 R4]^2) * \text{scale}$

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						where [] indicates IEEE32 representation NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Multiply Registers 32-bit (L:P:49)	MV/C M	REAL	3 or 4 registers	Either (optional)	Generic	Result is: $[R1R2] * [R3(R4)]$, meaning Regs 1 and 2 are a 32 bit number. The number is multiplied by Reg 3 (if 16 bit) or Reg 3 and 4 (32 bit number) 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/C M	REAL	1 register	No	CM4	Returns the IEEE power factor.
PM8 Power Factor IEEE (L:P:51)	MV/C M	REAL	1 register	No	PM8	Returns the IEEE power factor (converted from IEC mode as necessary). 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. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
SP40 Power Factor IEEE (L:P:52)	MV/C M	REAL	1 register	No	SEPAM 40	Returns the IEEE power factor (converted from IEC mode). NOTE: This logic code (as with 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/C M	REAL	2 registers	No	ML	Returns the IEEE power factor (converted from IEC mode as necessary). The second input register must be the associated Reactive Power for the Power Factor requested. NOTE: This logic code (as with 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/C M	REAL	2 registers	No	Generic	$R2/\sqrt{R2^2 + R1^2}$ where: R2 = real power R1 = reactive power NOTE: This logic code (as with 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/C M	REAL	4 registers	No	Generic	$[R3 R4] / \sqrt{([R3 R4]^2 + [R1 R2]^2)}$ where: R3 = real power IEEE32 MSR R4 = real power IEEE32 LSR R1 = reactive power IEEE32 MSR R2 = reactive power IEEE32 LSR

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
SP2000 Power Factor IEEE (L:P:56)	MV/C M	REAL	1 register	No	SEPAM 2000	Returns the IEEE power factor (converted from IEC mode). NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Scaled Register Signed 64-bit (L:P:57)	MV/C M	REAL	4 registers	Either (optional)	Generic	Reads a 64-bit signed integer and returns a REAL value. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Power Factor IEEE (L:P:58)	MV/C M	REAL	2 registers	No	Generic	Takes a 4 quadrant power factor (IEEE32 real) and returns an IEEE power factor
IEEE 64-bit double (L:P:59)	MV/C M	REAL	4 registers	No	Generic	Uses the IEEE standard for floating-point arithmetic (IEEE 754); returns the value as 32-bit REAL. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
IEEE 64-bit double (L:P:60)	MV/C M	STRING	4 registers	No	Generic	Uses the IEEE standard for floating-point arithmetic (IEEE 754); returns the value as 64-bit STRING. NOTE: This logic code (as with all REAL logic codes) has an accuracy of 15 digits. Anything longer than 15 digits should not be considered accurate.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
WRITES (these are write-only; see below for Read/Write codes)						
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 wherever 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 wherever there is a 1 in the mask and leaves the rest alone).
Write Register Unsigned (L:P:110)	MV/C M	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. 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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
Write Register Signed (L:P:111)	MV/C M	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. 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/C M	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.

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.
- **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.

The Profile Wizard

This chapter discusses how to use the Profile Wizard to add an individual device profile to, or remove it from, a project. From the Profile Editor, you export profile data into a file that is available to be imported into a PowerSCADA Expert project. From there, use the Profile Wizard to add the device profile into the project. Real-time tags, control tags, and alarm tags in the Profile Editor become variable tags in PowerSCADA Expert. 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 PowerSCADA Expert 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 [Add Multiple Devices through the Automation Interface on page 102](#) 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:

[Before You Use the Profile Wizard on page 87](#)

[Add the Cluster, Network Address, and Servers on page 88](#)

[When You Set Up More than two I/O Servers per Cluster on page 90](#)

[Using the Wizard to Add and Remove Devices on page 91](#)

[Add a Device to a Project on page 92](#)

[Add a TCP Device to a Project on page 92](#)

[Add a Serial Device to a Project on page 94](#)

[Add a DNP3_TCP Device on page 96](#)

[Add an IEC 61850 Device to a Project on page 98](#)

[Port Names on page 100](#)

[Remove a Device From the Project on page 101](#)

[Edit Devices in a PowerSCADA Expert Project on page 100](#)

[Add Multiple Devices through the Automation Interface on page 102](#)

[Use the OLE Automation Interface to Add and Remove Devices on page 103](#)

[Examples on page 104](#)

[Command Line Interface on page 105](#)

[Interface Workflow on page 105](#)

Before You Use the Profile Wizard

Before you can add device profiles, you need to establish the folder structure and network addresses in the project: Launch the product: click Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 > PowerSCADA Expert 8.0 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 (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).
- 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 Control Clients.

- a **project** to PowerSCADA Expert: 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 PowerSCADA Expert.

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 [Set a New Page as the Project Startup Page on page 112](#).

NOTE: When running the Computer Setup Wizard: If you choose Multi-Process (Server and Control Client), you must also choose **Networked (connect to other SCADA computers)** at the following screen. Otherwise, reporting functionality may be affected.

Add 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 scalability) helps 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.
2. From the Project Editor screen, click Servers > Clusters.

3. At the Cluster screen, type the cluster name.

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 automatically chooses it for a project. If there are multiple clusters, you have to select the one you want.
4. If desired, type a description in the Comment field.

This field can be used for translation purposes. See [Enter Copy for Translation on page 91](#) 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 and click Add to save the new name. Both clusters will be in the system. To view all of the clusters added to this project, use the scroll bar on the 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 [Enter Copy for Translation on page 91](#) 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:
 - 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, specify whether this server is primary or standby. The standby server acts 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 other server types that you need to add.

NOTE: When you delete a cluster, you must 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:

- 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 are removed.
- For additional information, click Help at the error screen.

When You Set 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 [Add the Cluster, Network Address, and Servers on page 88](#)). 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:

- 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).

- 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 switches to a redundant I/O server, affected devices will momentarily lose communication 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.

Enter 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\PowerSCADA Expert 8.0\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.

Using the Wizard to Add and Remove Devices

This section describes how to use the Profile Wizard to manage device information in a project. Before you begin adding devices, make sure you have completed the tasks described in [Before You Use the Profile Wizard on page 87](#).

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

[Add a Device to a Project on page 92](#)

[Add a TCP Device to a Project on page 92](#)

[Add a Serial Device to a Project on page 94](#)

[Add a DNP3_TCP Device on page 96](#)

[Add an IEC 61850 Device to a Project on page 98](#)

[Port Names on page 100](#)

[Remove a Device From the Project on page 101](#)

[Edit Devices in a PowerSCADA Expert Project on page 100](#)

[Add Multiple Devices through the Automation Interface on page 102](#)

Add a Device to a Project

The Profile Wizard allows you to add one device at a time. To add more than one device at a time, use the Automation Interface. See [Add Multiple Devices through the Automation Interface on page 102](#) for instructions.

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

[Add a TCP Device to a Project on page 92](#)

[Add a Serial Device to a Project on page 94](#)

[Add a DNP3_TCP Device on page 96](#)

[Add an IEC 61850 Device to a Project on page 98](#)

Add a TCP Device to a Project

Use this procedure when you want to add a generic TCP or Modbus TCP device to a 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 [Add the Cluster, Network Address, and Servers on page 88](#)).

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 PowerSCADA Expert 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 I/O servers per cluster, you must set them up manually (see [When You Set Up More than two I/O Servers per Cluster on page 90](#)).
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.

Add 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 project. Click Next.
NOTE: To ensure that the Alarm Log displays properly: Use the correct PM5000S or PM5000S1 driver for devices:
Use the PM5000S driver (for the most recent Alarm Log implementation) with:
 - PM51XX
 - PM53XX
 - PM55XX
 - PM5350PB
 - PM5350IB with FW version 3.00 and higherUse the PM5000S1 driver (for previous Alarm Log implementation) with:
 - PM5350 with FW prior to version 3.00
5. At the Enter instance information screen, type a descriptive profile name, for example: *CM4Bay1Circuit1* (no spaces or punctuation; to allow space in PowerSCADA Expert, 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 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\PowerSCADA Expert 8.0\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.
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 [Port Names on page 100](#).)
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 PowerSCADA Expert screens, compile the project.
14. When all errors are corrected, click File > Run to view the runtime environment.

Add a Serial Device to a Project

Use this procedure when you want to add a generic serial or DNP3 via serial device to a 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 [Add the Cluster, Network Address, and Servers on page 88](#)).

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 PowerSCADA Expert 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 You Set Up More than two I/O Servers per Cluster on page 90](#)).
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 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 project. Click Next.

NOTE: To ensure that the Alarm Log displays properly: Use the correct PM5000S or PM5000S1 driver for devices:

Use the PM5000S driver (for the most recent Alarm Log implementation) with:

- PM51XX
- PM53XX
- PM55XX
- PM5350PB
- PM5350IB with FW version 3.00 and higher

Use the PM5000S1 driver (for previous Alarm Log implementation) with:

- PM5350 with FW prior to version 3.00

4. At the Enter instance information screen, type a descriptive profile name, for example: *CM4Bay1Circuit1* (no spaces or punctuation; to allow space in PowerSCADA Expert, 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 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\PowerSCADA Expert 8.0\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, and so on). 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 [Port Names on page 100.](#))
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.

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 screens, compile the project.
13. When all errors are corrected, click File > Run to view the runtime environment.

Add a DNP3_TCP Device

Use this procedure when you want to add a DNP3_TCP device to a 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 [Add the Cluster, Network Address, and Servers on page 88](#)).

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 PowerSCADA Expert 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 You Set Up More than two I/O Servers per Cluster on page 90](#)).
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.

Add 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 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 PowerSCADA Expert, 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 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\PowerSCADA Expert 8.0\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 [Port Names on page 100](#).)
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.

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

Add an IEC 61850 Device to a Project

Use this procedure when you want to add an IEC 61850 device to a 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 [Add the Cluster, Network Address, and Servers on page 88](#)).

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 PowerSCADA Expert 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 You Set Up More than two I/O Servers per Cluster on page 90](#)).
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.

Add 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 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 PowerSCADA

Expert, 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 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\PowerSCADA Expert 8.0\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 screens, compile the project.
9. When all errors are corrected, click File > Run to view the runtime environment.

Port Names

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 [Add a Device to a Project on page 92](#) for more information.

Edit Devices in a PowerSCADA Expert Project

Edit a Profile and Add it Back to the Project

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

After you have added devices to the 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 project.

NOTE: If you have made manual changes to the profile in PowerSCADA Expert (Citect), do not use this process: you could corrupt your data. You must delete the

device from the 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:

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 PowerSCADA Expert, 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.

Edit a Device in PowerSCADA Expert Only

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

1. Delete the device from the 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 project.

Add Device Data in PowerSCADA Expert Only

If you need to add a small amount of data to a device that is in the project (e.g., add a single tag), add it directly in PowerSCADA Expert. 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.

Remove a Device From the 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.

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.

Add 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 project. (This example is for MODBUS TCP. See the table at the end of these instructions for other comms methods.)

NOTE: If you install the product 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 PowerSCADA Expert. If Excel is installed after PowerSCADA Expert, you will see an "out of memory" message when you try to run the macro. The solution is to uninstall both PowerSCADA Expert and Excel. After that, first reinstall Excel, then reinstall PowerSCADA Expert.

1. For an existing project: Before you begin, make a backup copy of your project. See [PowerSCADA Expert Backup on page 143](#) 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 87](#)).
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\PowerSCADA Expert 8.0\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 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\PowerSCADA Expert 8.0\Config.
 - b. Cell B3: Type the project name.
 - c. Cell B4: Enter the path to the location of *equipment.profiles*:
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), for example:
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 PowerSCADA Expert 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 PowerSCADA Expert 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	Data Bits	Stop Bits	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

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

2 Optional

Use the OLE Automation Interface to Add and Remove Devices

In this section, you will find these topics:

[Examples on page 104](#)

[Command Line Interface on page 105](#)

[Interface Workflow on page 105](#)

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 **IEquipmentAutomation** 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\PowerSCADA Expert 8.0\Config\citect.ini";  
  
myAuto.ReadProfiles(  
@"C:\Documents and Settings\All Users\Application Data\Schneider  
Electric\PowerSCADA Expert 8.0\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");  
        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 "master" device profile XML file. The given file must conform to the profiles schema.

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

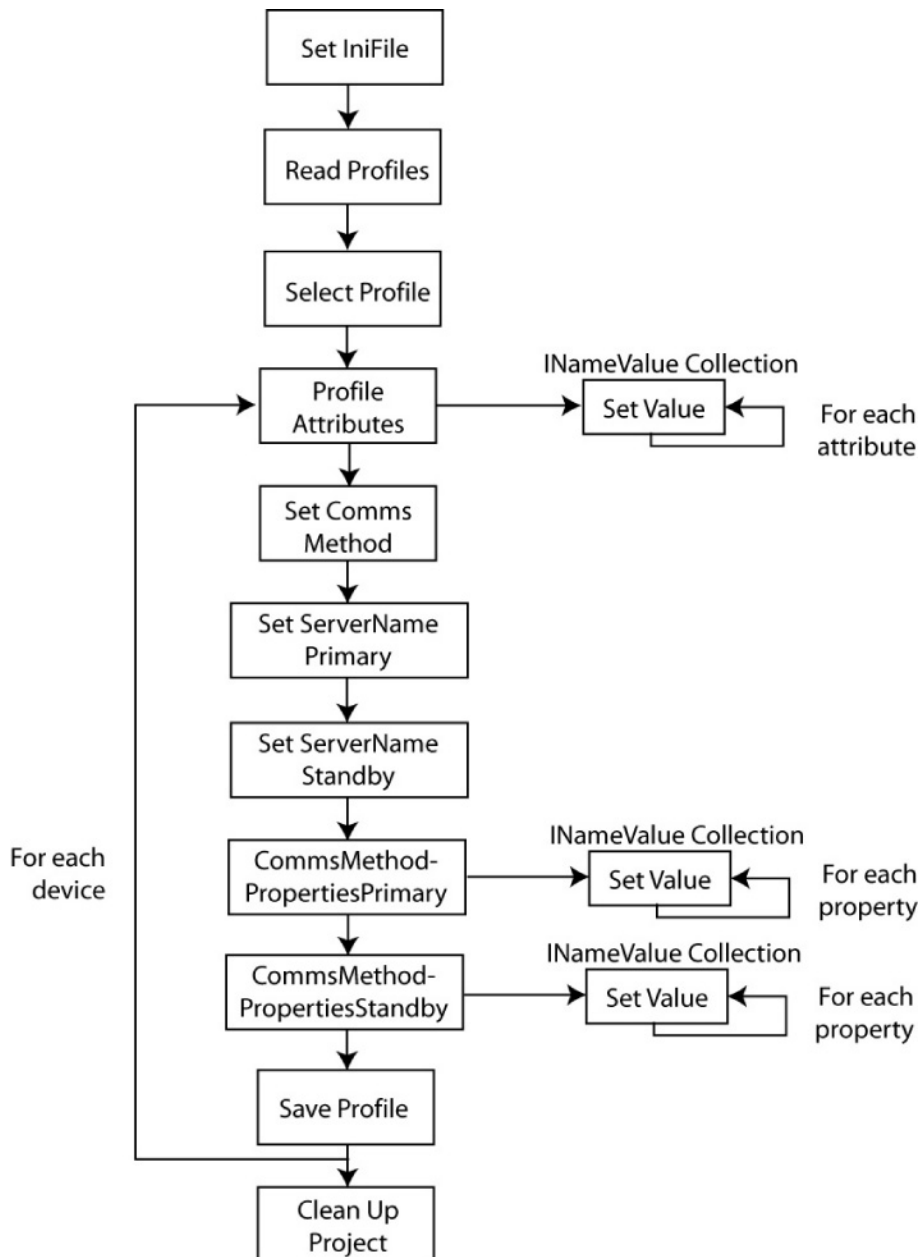
```
EquipmentWizard.exe /?
```

```
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.



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.

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 matches the expected schema.

SelectProfile—Select a profile to initialise 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 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.

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:

[Typical Workflow on page 109](#)

[The PowerSCADA Expert Pages on page 110](#)

[Create a Project on page 110](#)

[Use PowerSCADA Expert One-Lines](#)

[One-Line Flowchart on page 118](#)

[What Are PowerSCADA Expert One-Lines? on page 120](#)

[One-Line Device \(zOL\) on page 120](#)

[One-Line Colors on page 121](#)

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#)

[Start and Stop One-Lines on page 123](#)

[Work with Genies on page 123](#)

[Work in the Configuration Utility on page 128](#)

[Work with Alarms on page 138](#)

[Alarms Overview on page 138](#)

[Work in the Configuration Utility on page 128](#)

[Add Setpoints and Delays on page 139](#)

[Set Up an Alarm Based on an Enumeration on page 139](#)

[Change an Alarm Severity on page 140](#)

[Enable Waveforms for Onboard Alarms on page 140](#)

[Set Parameters for Event Log Length and Historical Logging of Events on page 140](#)

[Add an Onboard Alarm Tag on page 141](#)

[Set Up Audible Alarms on page 142](#)

[Restore and Back Up Projects on page 142](#)

[Restore a Project on page 142](#)

[PowerSCADA Expert Backup on page 143](#)

[Use Cicode to Customize a Project on page 143](#)

[Work in the Configuration Utility on page 128](#)

[Work in the Configuration Utility on page 128](#)

[Trend Tag Scan Intervals on page 147](#)

[Disk Storage Calculation for Trends on page 148](#)

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 PowerSCADA Expert; 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 the project

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 [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 115 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.

The PowerSCADA Expert Pages

When you launch the product (v8.0), three windows are opened:

- **Citect Explorer:** Use this application to explore and add projects. The product 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 PowerSCADA Expert configuration. For example, use it to add clusters, network addresses, and servers; and to launch the Profile Wizard and the One-Line Configuration utility.
- **Citect Graphics Builder:** Use this application to build graphic display pages for a project. A standard PowerSCADA Expert library of genies is 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.

Create a 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 [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 115.

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 PowerSCADA Expert.

In this section, you will find these topics:

[Reuse Projects Created in the Profile Editor](#) on page 111

[Add a New Graphics Page](#) on page 111

[Set a New Page as the Project Startup Page](#) on page 112

[Create a One-Line on a Graphics Page](#) on page 112

[Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 115

[Create New Genies](#) on page 116

[Delete Information from PowerSCADA Expert](#) on page 118

Reuse 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 server computer, paste Equipment.profiles in this location, where:
[Drive Letter]: is the drive on which you installed the 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 PowerSCADA Expert (see Before You Export, above).
5. Be sure you have created the files described in [Before You Begin on page 111](#).

Before You Begin

Before you start adding data in the 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; from the Citect Editor, added clusters, network addresses, and servers
- exported devices from the Profile Editor
- added devices into the PowerSCADA Expert project, using the Profile Wizard

Add a New Graphics Page

You need to add a project in two places before you add pages. First add the project to PowerSCADA Expert, from the Citect Explorer screen (see steps below). Then add it to the Profile Editor. See [Add, Edit, or Delete a Project on page 52](#) 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 PowerSCADA Expert 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 [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\) on page 115](#) 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.

Change the Background Color of Pages

You have the option of changing the background color of individual pages in your project. To do this, follow these steps:

1. Create a new project, using "high contrast" as the starter project.
This project will already include all of the pages that you will use. There will also be a new parameter, called *PageColorExt*. This parameter includes the extension that allows you to color backgrounds.
2. To designate background color for each page, pick the color in the Graphics Editor.

You can also change the genie font colors on your project pages. To do this see [Change the Genie Color in Project Pages on page 112](#).

Change the Genie Color in Project Pages

You have the option of changing the color of genies in pages of your project. To do this:

In your project, add a new parameter to the Graphics section, called *GenieFont*. Enter the value designation for the color you want to use. Make sure that there is sufficient contrast between the two colors so that the genie colors will be visible.

NOTE: If you use the "high contrast" starter project to create your project, this parameter is automatically added, as are all of the pages that you will use. For instructions on changing the background color of your project pages, see [Change the Background Color of Pages on page 112](#).

Set 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>.

Create 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 (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

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 PowerSCADA Expert features are available, choose a library that begins with “pls”). If you want to edit a genie or create a new one, see *Create New Genies* 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 for the appropriate object:
 - Configure a Busbar* on page 125
 - Configure a Circuit Breaker or Switch* on page 125
 - Configure a Meter* on page 126
 - Configure a Source* on page 126
 - Configure a Transformer* on page 127
- NOTE:** An asterisk in any of the windows indicates that it is a required field.
5. 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.
 6. 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.
 7. On the Graphics Page, click File > Compile.

While the page is being compiled, the Compiler displays each file name as it is being compiled. A message then displays, telling you that compilation was successful.
 8. 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 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.
 9. 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).
 10. If you delete any entries, click File > Pack to clean up the files.
 11. 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).
 12. 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.

13. 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 [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 115 for help.

Enable Lockout/Tagout



⚠ DANGER

HAZARD OF ELECTRIC SHOCK, EXPLOSION, OR ARC FLASH

- Do not rely solely on the display of the icon on the one-line. Before working on the equipment, verify that the device is physically locked out/tagged out before performing work on it or any downstream equipment.
- Ensure that all safety regulations and procedures have been followed before you work on the equipment.

Failure to follow these instructions will result in death or serious injury.

NOTE: Do not incorrectly configure the tag, as this can lead to unexpected equipment operation. Also consider the possibility of communications loss that could yield false readings.

With this feature, you can cause the "locked out" icon (shown above) to display on your graphics page in runtime. The icon displays when a tag attribute for a device reaches a specified value. For example, you might set a PLC tag to 0 when the equipment is in lockout/tagout (the door is open), and to 1 when the equipment status indicates that the door is closed.

This is a read-only feature; but it does not prevent controls to the device or area. This feature is not available in PLS_Example.

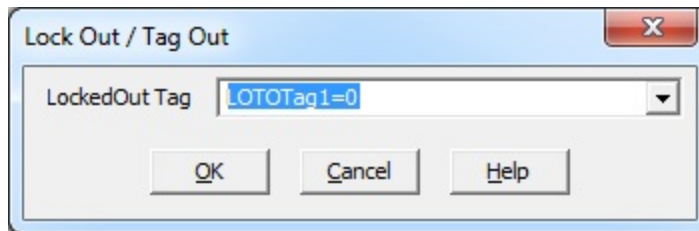
Follow these steps to enable the icon for a device:

1. From the pls_gen_eq_2 genie library, add the lockout/tagout genie to the graphics page. Position it beside the equipment that is being monitored.
2. From Variable Tags, choose the device tag (or create a custom tag) that you want, and associate it with the device that will read the lockout/tagout status.
3. Define what indicates the status (for example, *door status open*, 0 = false and 1 = true).
4. Edit the lockout/tagout genie to read the tag on the device. Make the lockout/tagout genie visible when the device tag indicates that the device is in lockout/tagout status; and make the icon hidden when the device is not in lockout/tagout. (By default, lockout/tagout status is hidden when the tag property is "NOT [variable tag].")

To continue the example, if the tag is reading *door status open* and the value = 0, then the door is open and in lockout/tagout; if the value = 1, the door is closed and

not in lockout/tagout.

The following graphic illustrates how it might look in when adding the genie:



Use Menu Configuration to Edit Pagemenu.dbf (Change 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 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):

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 maximum)
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 maximum)

Menu Item	Description
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: PLSPageDisplay("CB_IEC_1") which displays the page CB_IEC_1.
4: Symbol	Displays a defined image along with the description for that level. 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 . 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.

Create New Genies

There are two ways to add new one-line genies to a project:

Copy a genie: Create a genie that is completely compatible with current genies, having the same genie parameters and functionality. To do this, you can copy an existing genie and change attributes. This type of genie must be added to the [SupportedGenies.xml File on page 127](#).

Create a unique genie: Create a genie that has unique parameters, validation requirements, and output types. See instructions below for this. This type of genie must be added to both the [SupportedGenies.xml File on page 127](#) and the [GenieConfiguration.xml File on page 127](#).

The easiest way to create a new genie is to make a copy of a similar genie from the standard 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 PowerSCADA Expert genies, including all of the smaller parts that you could use to create genies, see [Appendix 1: Default Genie Library on page 212](#).

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

To create a copy of a genie:

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:

PowerSCADA Expert genies are complex genies with the added ability to animate color during runtime (showing changing states). These genies are comprised of multiple symbols that have the same attributes (all call the same Cicode function).

1. Create the new symbol.
2. Configure the symbol to display the customized colors in the runtime environment by calling the *PLS_GetBusColorIndex(INT nColorIndex)* function, which is provided in PowerSCADA Expert. This function must be called from the Array Condition area of the Symbol properties window.

Each Array symbol can display up to 255 colors. Please ensure that the default color palette matches the palette used in PLS_Include. The parameter for *PLS_GetBusColorIndex(INT nColorIndex)* is the busbar number that determines the animated color. Always use the %variable% notation. Using this notation permits you to reuse genies.

See the default symbols in the PowerSCADA Expert libraries that begin with PLS_.

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

Set Up IEC 61850 Advanced Control

The advanced control window provides two advanced controls (synchro check and interlock check) that you can use with IEC 61850 IEDs.

See [Perform IEC 61850 Advanced Control on page 167](#) for information on using these advanced controls.

WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not incorrectly configure the tag, as this can lead to unexpected equipment operation.
- Ensure that you understand the effects of using the "bypass" option so you do not shut down critical equipment.
- Also consider the possibility of communications loss that could yield false readings.
- Ensure that all safety regulations and procedures have been followed before you work on the equipment.

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

Enable the Advanced Control

Before you can use the advanced control, you must add the appropriate variable STRING tag to be used when you send the command. For breaker control, the "operate" tag typically used is:

```
S33K_A_INC\CSWI1\Pos\ctVal
```

For this tag, you need to then add the corresponding STRING tag:

```
S33K_A_INC\CSWI1\Pos\ctVal\str
```

If you are using select before operate (SBOw), you also need to add a STRING tag for it.

Delete Information from PowerSCADA Expert

If you find you need to delete any data that you have entered (clusters, servers, genies, and so on), 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.

Use PowerSCADA Expert One-Lines

Click the following links for related one-line information.

[One-Line Flowchart on page 118](#)

[What Are PowerSCADA Expert One-Lines? on page 120](#)

[One-Line Device \(zOL\) on page 120](#)

[One-Line Colors on page 121](#)

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#)

[Start and Stop One-Lines on page 123](#)

One-Line Flowchart

This chart provides an overview of the process that you will follow to set up and use animation in one-lines.

For detailed information on one-lines see:

Running the AOL Configuration Utility:

[Review Genie Configurations on page 128](#)

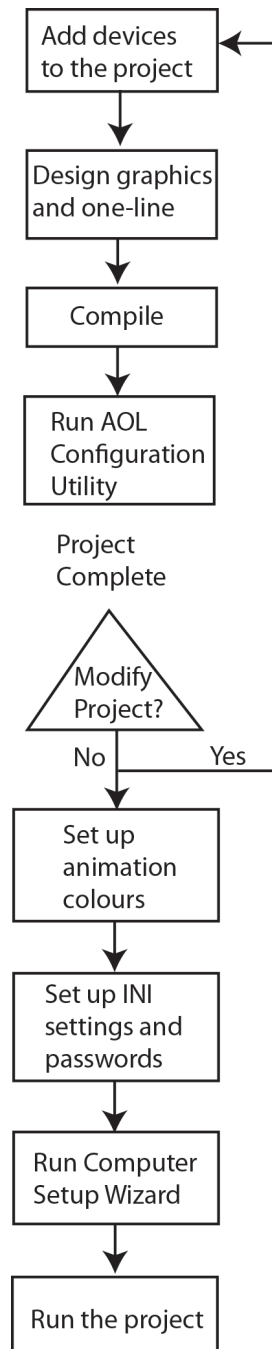
[Perform Repairs for One-Lines on page 131](#)

Setting Up Animation Colors:

[One-Line Colors on page 121](#)

INI Settings and Passwords:

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#)



After you run the project, ensure that the password is encrypted (see *IsEncrypted* in [Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#)).

Ensure that all steps in the Quick Start procedure have been followed: [Quick Start: Typical Workflow on page 5](#).

What Are PowerSCADA Expert One-Lines?

PowerSCADA Expert one-lines provide the following features. They:

- graphically propagate unknown conditions downstream through a one-line diagram. When there is a communications loss to a device, the one-line provides accurate system status, even though information is not available.
- eliminate "seesaw logic": when bus status temporarily toggles on and off after an upstream breaker opens
- handle the continuation of a one-line diagram when it extends from one page to another
- provide built-in support for power flow diagrams; one-line color "animates," based on the source that is feeding the circuit.

Where are the files located?

Listed below are several important files in one-line animation. Changing any of these files could make the animation be inoperative.

PLS_Include	
AdvOneLine.exe	Application running the core one-line bus animation logic engine
AdvOneLine.ini	Specifies performance and configuration parameters for AdvOneLine
AdvOneLine.csv	One-line example configuration
Current RUN project	
AdvOneLine.csv	Specifies the system configuration for AdvOneLine
AdvOneLine.ini	Specifies performance and configuration parameters for AdvOneLine
AdvOneLineDebugBus.csv	Created at runtime to help debug the AdvOneLine.csv by specifying all bus connections
AdvOneLineStatusLog.txt	Created at runtime to specify status and error messages that might occur

One-Line Device (zOL)

To use one-line graphics, you must create a memory device named zOL. One-line graphics use the zOL device to drive animation. You must have at least one zOL device per project.

To add the zOL device:

1. Open the Profile Wizard.
2. Click *Create an I/O Device in the project*. Click *Next*.
3. Select the device called *OneLine Device Setup*. Click *Next*.
4. Follow the remaining steps to add the device.

By default, this device will support 100 sources, 1000 buses, 1000 meters, and 1000 breakers. You can modify this via the Profile Editor:

1. On the Setup Projects tab, choose the project.
2. Click the Project Parameters subtab.

3. Enter the optional project parameters (MaxBreakers, MaxBuses, MaxMeters, MaxSources). Valid entries are from 1 to 9999 (only 200 for MaxSources).
4. On the Selected Device Profiles subtab, click Refresh Tags.
5. Export the project.
6. In the Profile Wizard: Select the *Update one or more I/O Device(s) in the project* option. This updates the zOL I/O device parameters entered in step 3.

The new one-line device is ready to be used in the selected project.

One-Line Colors

In previous versions of the software, line coloring was determined by the line active state of the busbar. Driven by the busbar color, this state was then fed to all of the components on that page.

In PowerSCADA Expert 8.0, coloring is based on the source and meter line active states. Sources dictate the colors for each genie. Meters can only determine if a bus is active. When the bus is live, the meter then colors based on the source that is connected to the bus. If there is no source, the default color is used.

NOTE: Depending on how you configure transformers, you can either use this "pass-through" coloring, or you can use "voltage-level" coloring. See [Configure a Transformer on page 127](#) for more information.

Assigning Colors

To assign a color to a source, open the Color Configuration tab of the One Line Configuration Utility (Citect Editor > Tools > One-Line Configuration).

1. Choose the project for which you want to assign colors.
2. Choose the Project Color Palette. Select the project in which the project genies are defined: usually PLS_Include.
3. For each source or transformer, choose the desired color: click the color cell for that source/transformer; then choose the color from the dropdown list. You can also select a color for unknown sources, off, and error. To indicate a flashing color, you can select two colors.
4. When all colors are assigned, click Save.

Add INI Settings to AdvOneLine.ini and Citect.ini

NOTE: See [Application Configuration Utility on page 203](#) for screens that streamline configuration.

All .ini settings are made in the OneLineEng section of the .INI files.

You must have correct .INI settings in order for the one-lines to run properly. Ensure that the following .INI parameters are properly set:

AdvOneLine.ini Settings

The following parameters are set in the Application Configuration Utility ([One-Line Engine on page 206](#)): UpdateInterval, PrimaryServerIP, StandbyServerIP, HealthTimeout, MaxStartupDelay, LoginUserName, LoginPassword, and LogFileLength.

Parameter	Description	Default Value
UpdateInterval	The interval at which the system tries to solve the system one-line. This interval can be changed to slow down the rate at which the animation is solved. Specifying a rate faster than possible will force the engine to solve the system as quickly as possible.	1000 msec
PrimaryServerIP	Used by redundant configurations to specify the IP address of the server on which the primary IOMServer resides. This parameter is required for a redundant configuration. If either the primary or standby IP addresses are not specified, the logic engine will assume that the system is not redundant.	N/A
StandbyServerIP	Used by redundant configurations to specify the IP address of the server on which the standby IOMServer resides. This parameter is required for a redundant configuration. If either the primary or standby IP addresses are not specified, the logic engine will assume that the system is not redundant.	N/A
HealthTimeout	This is a performance parameter that dictates that amount of time that must elapse before the one-line engine is considered non-functioning, and a PC-based alarm is raised in PowerSCADA.	[UpdateInterval] + [TagSubscribeWait] * 5 milliseconds Minimum value: 1000 msec
DefaultColor	This cosmetic parameter tells the engine the default color to be assigned to objects on the screen at system startup. This is useful for identifying components that have been left out of the CSV or simply as a means of having the engine set all currently unused objects to a color that indicates that they are not being monitored. If an invalid color is specified, the engine will default to black.	250
MaxStartupDelay	Sets the amount of time the AdvOneLine.exe has to start up. If this time is exceeded, initial tag subscriptions will not succeed, and the EXE will report an exception.	60 sec
StartupDelay	Sets the amount of time after AdvOneLine.exe has started for the system to be on line and all initializations complete.	[UpdateInterval] + [TagSubscribeWait] * 5 milliseconds Minimum value: 1000 msec
LoginUserName	This is the Citect user name to be used for the ctAPI connection in AdvOneLine.	aol
LoginPassword	This is the Citect user password to be used for the ctAPI connection in AdvOneLine.	aol
IsEncrypted	Determines if the password is encrypted. The first time the project is opened in run time, the password is automatically encrypted, and this will be set to True.	False (changed to True after the first run and successful password encryption)
CitectIniPath32	Provides the path to the global Citect.ini file for a 32-bit operating system install. This setting must be changed if SCADA is not installed on the C: drive, or if the Citect.ini file is moved/installed in another directory.	Default value: C:\Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.0\Config
LogFileLength	Suggests the log file length in number of lines. After surpassing this limit, the log file is saved with suffix ".bak," and a new file is created.	Default value: 5000 Allowed values: 10–10000
DebugLevel	Sets the level of logging for AdvOneLine.exe. Multiple values are separated by (e.g., Error Warn).	N/A Allowed values: All, Error, Warn, Debug

Citect.ini Settings

Parameter	Description	Default Value
AutoRestart	Indicates whether the one-line will restart itself when the logic engine is not responding.	0 (disabled) Allowed values: 0, 1
ServerRole	Informs the local instance of PowerSCADA where it is (primary or standby server). This parameter is controlled by the AdvOneLine.exe application. The user does not need to create or modify this parameter. It is set based on the PrimaryServer IP and StandbyServerIP parameter settings.	Primary
StartupDelay	This is a performance parameter that dictates that amount of time that must elapse before the one-line engine is considered nonfunctioning, and a PC-based alarm is raised in PowerSCADA.	15 sec

Start and Stop One-Lines

If you need to start or stop AdvOneLine.exe, call one of these Cicode functions.

Only call these functions on an I/O server that is communicating with an [One-Line Device \(zOL\)](#) on page 120.

To stop AdvOneLine.exe, call:

```
PLS_StopAdvOneLine(STRING sIOServer="",STRING sCluster="")
```

To start AdvOneLine.exe, call:

```
PLS_StartAdvOneLine(STRING sIOServer="",STRING sCluster="")
```

NOTES:

- If the default parameters are used, the functions will run on the local machine.
- If you call the function from a remote server, enter the I/O server name and cluster to run the function on that server. You must be logged in to perform this action.

Work with Genies

Use these topics to create and configure genies:

[Genie Type Descriptions](#) on page 124

[Configure an Automatic Transfer Switch \(ATS\)](#) on page 124

[Configure a Busbar](#) on page 125

[Configure a Circuit Breaker or Switch](#) on page 125

[Configure a Meter](#) on page 126

[Configure a Source](#) on page 126

[Configure a Transformer](#) on page 127

[Enable Lockout/Tagout](#) on page 114

[Create New Genies](#) on page 116

[GenieConfiguration.xml File](#) on page 127

[SupportedGenies.xml File](#) on page 127

Genie Type Descriptions

In the One Line Configuration Utility, there are several genies. Each one has a certain defined list of information that displays by default. You can also expand this list to include everything that is known about that genie type. To learn about each genie's data, click one of the following links:

[Breaker and Switch Information on page 134](#)

[Meter Information on page 135](#)

[Source Information on page 136](#)

[Transformer Information on page 137](#)

For configuration instructions, see:

[Configure a Busbar on page 125](#)

[Configure a Circuit Breaker or Switch on page 125](#)

[Configure a Meter on page 126](#)

[Configure a Source on page 126](#)

[Configure a Source on page 126](#)

[Configure an Automatic Transfer Switch \(ATS\) on page 124](#)

See also:

[Enable Lockout/Tagout on page 114](#)

[GenieConfiguration.xml File on page 127](#)

[SupportedGenies.xml File on page 127](#)

Configure an Automatic Transfer Switch (ATS)

Use this option to configure automatic transfer switches (ATS) that you add to a one-line in the graphics builder.

1. Display the Transfer Switch window by pasting the appropriate genie on to the graphics builder page.

Transfer Switch Information: Left/Right/Bottom

2. **Breaker Number:** Type the ID numbers you want to use for the left side and right side of this ATS. Valid entries: 1–1000. Each number must be unique within this one-line.
3. **Busbar:** Type the numbers of the left and right source busbars, and for the destination (bottom) busbar. Valid entries: 1–1000.
4. **Label:** For each ATS side, type the information that you want to appear on the switch in the runtime environment (example: *Pri* and *Emer*).
5. **Closed Expression:** For each side of the switch, type the information that should display when that side is closed (example: *Tag1* <> 1). Do not use "NOT" in the expression.

Display Information

1. **Label 1/Label 2:** Type the descriptive information that you want to appear in the upper left corner of the genie in the runtime environment.
2. Click **OK** to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 121](#).

Configure a Busbar

Use this option to configure busbars that you add to a one-line in the graphics builder. Using this option, you can assign a busbar number to a busbar, and ultimately, to use busbars to connect genies on a one-line.

Busbar numbers associate devices within drawings, and they help you set up animation for genies. The busbar entered here is also used for the associated device.

1. Display the busbar window by pasting the busbar genie on to the graphics builder page.
2. **Busbar Number:** type the number you want to use for this busbar. Valid entries: 1–1000.
3. **Line Active:** This field is no longer used.
4. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 121](#)

Configure a Circuit Breaker or Switch

Use this option to configure circuit breakers or switches that you add to a one-line in the graphics builder.

1. Display the circuit breaker or switch window by pasting the appropriate genie on to the graphics builder page.
2. **Equipment:** From the drop-down list, choose the name of the equipment represented by the genie (this is the equipment name that you entered in the Profile Wizard or Automation Interface, e.g., *CM4Main*). To view the equipment that is available for this page, you must have the project selected in the Citect Explorer.
3. **Breaker/Switch Number:** Type the number you want to use for this breaker or switch. Valid entries: 1–1000. This number must be unique within this one-line.
4. **Source** and **Destination:** When you connect to busbars, type the numbers for the connection source and destination busbars. Valid entries: 1–1000.
5. **Label 1** and **Label 2:** Type the information that you want to appear in the upper left corner of the genie in the runtime environment.
6. **Value Type:** From the drop-down list, choose the tag you want to use for this genie. The related information displays in the lower left corner of the genie in the runtime environment.

This tag causes real-time data (such as *currents*) to display on the genie status page in the runtime environment. If you do not choose a tag here, the status page will not display the real-time data. See [Use the Equipment Pop-Up Page on page 164](#) for more information.

7. **Units:** From the drop-down list, choose the unit that you would like to display on the genie in the runtime environment. Units that display here were added in the Profile Editor (see [Add or Edit a Base Engineering Unit or Conversion on page 22](#)).
8. **Multiplier:** Enter the multiplier that is to be applied to the units chosen in step 7.
9. Click OK to save the genie to the page and to return to the graphics builder page.

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.

For information on how colors are determined, see [One-Line Colors on page 121](#).

Configure a Meter

Use this option to configure meters that you add to a one-line in the graphics builder.

1. Display the meter window by pasting the meter genie on to the graphics builder page.
2. **Equipment:** From the drop-down list, choose the name of the equipment represented by the genie (this is the equipment name that you entered in the Profile Wizard or Automation Interface, e.g., *CM4Main*). To view the equipment that is available for this page, you must have the project selected in the Citect Explorer.
3. **Meter Number:** Type the number you want to use for this meter (valid entries: 1–1000). This number must be unique within this one-line.
4. **Busbar Number:** Type the number for the busbar that connects to the meter. Valid entries: 1–1000.
5. **Line Active:** Enter the appropriate Cicode expression (such as *MyTag1 > 0*) to determine when the meter detects power on the busbar. See "How do I create and configure busbars" in [Appendix 5: Frequently Asked Questions \(FAQs\) on page 254](#) for more information.
6. **Label 1** and **Label 2:** Type the information that you want to appear in the upper left corner of the genie in the runtime environment.
7. **Value Type:** From the drop-down list, choose the tag you want to use for this genie. The related information displays in the lower left corner of the genie in the runtime environment.

This tag causes real-time data (such as *currents*) to display on the genie status page in the runtime environment. If you do not choose a tag here, the status page will not display the real-time data. See [Use the Equipment Pop-Up Page on page 164](#) for more information.

8. **Units:** From the drop-down list, choose the unit that you would like to display on the genie in the runtime environment. Units that display here were added in the Profile Editor (see [Add or Edit a Base Engineering Unit or Conversion on page 22](#)).
9. **Multiplier:** Enter the multiplier that is to be applied to the units chosen in step 8.
10. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 121](#).

Configure a Source

Use this option to configure sources (e.g., utilities and generators) that you add to a one-line in the graphics builder.

1. Display the source window by pasting the generator or utility genie on to the graphics builder page.
2. **Source Number:** Type the number you want to use for this source (valid entries: 1–100). This number must be unique within this one-line.

The source number is used when you determine coloring for the one-line. For more information on this, see [One-Line Colors on page 121](#).
3. **Busbar Number:** Type the number for the busbar that connects to the source. Valid entries: 1–1000.
4. **Line Active:** Enter the appropriate Cicode expression (such as *MyTag1 > 0*) to dictate when the source powers the busbar. See "How do I create and configure busbars" in [Appendix 5: Frequently Asked Questions \(FAQs\) on page 254](#) for more information.
5. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 121](#).

Configure a Transformer

Use this option to configure transformers that you add to a one-line in the graphics builder.

1. Display the transformer window by pasting the appropriate genie on to the graphics builder page.
2. **Top and Bottom Source Number:** For each source, whether top or bottom, type a source number to control voltage-level coloring for the secondary side busbar. Valid entries: 1–100.

There are two possible configurations:

Pass-through coloring: If you leave these fields blank, the transformer will transfer the color that is assigned to the primary side (source) to the secondary side (destination) busbar. For example, if Source 3 feeds the source busbar of a transformer, and you leave this field blank, then Source 3 will also feed the destination busbar (and the Source 3 color will be used).

Voltage-level coloring: When you enter top and bottom source numbers, the transformer colors the one line based on this number. For example, if Source 3 feeds the top of the transformer, but you enter 5 for the bottom source, the transformer feeds the color from Source 5 to the destination (bottom) busbar.

3. **Source and Destination:** When you connect to busbars, type the numbers for the connection source and destination busbars. Valid entries: 1–1000.
4. **Label 1 and Label 2:** Type the information that you want to appear in the upper left corner of the genie in the runtime environment.
5. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 121](#).

GenieConfiguration.xml File

[Example](#)

Use the GenieConfiguration.xml file to define both completely new (unique) genies and those that have been copied and modified from an existing genie.

This file defines each genie in detail. It links fields with genie parameters names, defines validation, and defines how to export each genie for the one-line.

Some fields have limited possibilities. See the comments for each part of the XML file.

See also: [SupportedGenies.xml File on page 127](#)

SupportedGenies.xml File

[Example](#)

Use this file to define genies that support one-line coloring.

This file links genies in a library to a "genie type." In this file, you need to define the project name, library name, and genie name. The genie name may be "*": which will select all genies that library. You can exclude individual genies.

See also: [GenieConfiguration.xml File](#)

Work in the Configuration Utility

Use these topics to learn about how you use the Configuration Utility to review genie configurations before you compile a project. Then you can view errors and warnings, and repair and upgrade genies.

[One-Line Configuration Utility on page 128](#)

[Review Genie Configurations on page 128](#)

[One-Line Errors and Warnings on page 130](#)

[Perform Repairs for One-Lines on page 131](#)

[Automatic Transfer Switch \(ATS\) Information on page 132](#)

[Busbar Information on page 133](#)

[Breaker and Switch Information on page 134](#)

[Meter Information on page 135](#)

[Source Information on page 136](#)

[Transformer Information on page 137](#)

One-Line Configuration Utility

To access this screen, click Tools > One-Line Configuration from the Citect Editor screen. There are two tabs.

On the **Genie Configuration tab**, you can:

- view genie types, along with their states (normal, warning, error) and their properties
- repair genies that are part of a version 7.30 or later project:
 - corrects incorrect breaker, source, meter, transformer, and Sim source numbers
 - corrects invalid line active when a connected busbar has a valid line active
- repair and upgrade genies that are part of a project from a version earlier than 7.30
 - rennumbers ALL breaker, source, meter, and Sim source numbers
 - corrects invalid line active when a connected busbar has a valid line active
 - reassigns ALL valid busbars

See *Review Genie Configurations* for information about using this tab.

On the **Color Configuration tab**, you can assign colors to sources. see [One-Line Colors on page 121](#) for information about using this tab.

Modify AdvOneLine.csv

After you run the One-Line Configuration Utility, open the project to verify that the animation is working correctly. If the animation is not correct, repeat the process of running the utility and verifying out animation until all errors are corrected.

Review Genie Configurations

Use the one-line configuration utility to review genie configurations before you compile your project.

Errors (✖) and warnings (⚠). You must correct errors; otherwise, you may not be able to compile, and the animation will not work. Although you might not need to correct warnings, you should review them to ensure that their settings are correct. Note

that, as you hover over an error or warning icon, a tooltip tells you what is wrong with the genie.

Before you use the compile, do the following:

- If you are upgrading from an earlier version, run Update Pages in the Graphics Builder.
- Create the pages.

To launch the utility:

1. Make sure you are viewing the system for which you want to view information.
2. From the Citect Editor, click Tools > One-Line Configuration.

A message displays, telling you to that Citect will be hidden.

3. Click OK.

The first time you load the utility, a large system could take a couple of minutes to load. After that, it should load with a minute.

The utility displays with the following information:

Field	Description
Project	Default: the project selected in Citect Explorer
Show By:	Type: Information is sorted first by genie type, then by page. This option is useful when you want to see all genies of a certain type together, regardless of where they are in the drawing pages. Page: Information is sorted first by page, then by genie type. This option is useful when you want to see all genies on a certain page.
Advanced Properties	Check this box to view the basic information plus any additional information relevant to that genie type.
State Filters:	Check the individual boxes for how you want to view information. For example, you might only be interested in viewing genies that have error states. This option controls only the genie information in the right-hand pane.
Genie Types	Types are: breakers, busbars, meters, sources, and transformers
Genie Information	Columns of information display: In the Basic (default) view: the most used information If you click Advanced Properties, you see the basic information, followed by all of the information known about the genie(s) you are viewing.
Repair—Upgrade Project	Check this box to cause the repair feature to repair the entire project. Use this feature only to upgrade projects that are earlier than PowerSCADA Expert 8.0. This option repairs the entire project, renumbering all busbars, breakers, meters, duplicate Sim sources, and sources. Additionally, busbar line active states are used to determine meter and source line active states. DO NOT perform Repair—Upgrade Project more than once, and do not perform it on a PowerSCADA Expert 8.0 project.
Repair	This feature attempts to repair errors and warnings.

For specific information about each type of genie, click a link below:

[Breaker and Switch Information on page 134](#)

[Busbar Information on page 133](#)

[Meter Information on page 135](#)

[Source Information on page 136](#)

[Transformer Information on page 137](#)

One-Line Errors and Warnings

Typical one-line errors are:

- CSV formatting errors
- files required by the logic engine are locked or open in another process
- nonexistent tags are specified in CSV conditions
- not running the Computer Setup Wizard for the runtime project

Communication Errors

When communication errors occur, the object that has lost communications gives an "unknown" status, which is graphically represented in the one-line animation.

Objects in the one-line should be defined to display the communication errors as a different color. The errors are calculated using the quality of a tag. If a tag or point becomes invalid, it is assumed that the communication is also off line. When this occurs, the graphical objects (buses, breaker, and sources) should change to the pre-set "unknown status" color (the array position 255 in the graphic).

Error Logging

The most common errors in a CSV file are logged to the Run project in a file named AdvOneLineStatusLog.txt. The file can contain several messages. The table below lists these errors and their descriptions.

Error Message	Description
Main Execution Loop Unexpected Failure	The main logic loop has thrown an exception that has not been handled by other error messages.
AdvOneLineDebugBus.Csv is locked	Another process or user has this required CSV file locked. Ensure that you do not have the file open.
PowerSCADA ExpertRunning Project Path: "PATH" Does not Exist. Please Shutdown your Project and Try Running your Computer Setup Wizard	The Citect.ini "Run" parameter has an invalid project path that does not exist. Run the Computer Setup Wizard, and this path should be corrected.
PowerSCADA ExpertRunning Project Pth Not Specified. Please Shutdown your Project and Try Running your Computer Setup Wizard	This problem is almost exclusively caused by not running the Computer Setup Wizard.
PLSCADA is not in runtime	You must have your project running before you execute the AdvOneLine.exe file.
Failed to Establish Connection with CTAPI. PLSEngine.establishPLSConnection (FAILED CONNECTION)	This error message indicates the PLS API connection has unexpectedly been disconnected.
Required CSV file is locked	The CSV file specified (AdvOneLine.csv) is locked by another process or user. Ensure that you do not have the file open.
Invalid prefix located in CSVParser.FormatCSVData	The CSV parser has detected an invalid component prefix. This error message should not occur.
ERROR: Duplicate Component Name Encountered	Check the CSV file to ensure that you do not have two sources, meters, or breakers with the same component number.

Error Message	Description
ERROR: Invalid Node1 Number Encountered	In the Bus1 column, you have a node that is not a number between 1 and 1000.
ERROR: Invalid Node2 Number Encountered	In the Bus2 column, you have a node that is not a number between 1 and 1000.
ERROR: Node Not Specified	You have a component without a Bus1 and/or Bus2 specified.
ERROR: Invalid Condition String Encountered (MESSAGE)	You have a syntax error in your condition column. Read the message. It will give details about the syntax error, the line on which it occurred, and (if applicable) the character at which it occurred.
One or more of the tags specified in your CSV file do not exist in your Runtime SCADA project	Examine your CSV file. Either add the tags listed above the error message, or remove the tags from the CSV

By default, only exceptions are logged. For more details, see the information on the *Add INI Settings to AdvOneLine.ini and Citect.ini*.

Perform Repairs for One-Lines

Before you begin, back up your project.

CAUTION
DATA CORRUPTION Always make a backup copy of your project before you perform a repair. Failure to follow these instructions can result in data loss and corruption of your project.

Follow these steps to view the One Line Configuration Utility, and to make repairs to your projects:

1. Open the utility: From the Citect Project Editor, click Tools > One-Line Configuration.
The first time you launch this utility, it could take several minutes for it to read all of the project information.
The utility will read the project's graphics pages to analyse the project. If it appears that the project is from a version prior to PowerSCADA Expert 8.0, a message displays, telling you that the project appears to be out of date. The default "Upgrade Project" is checked in the lower right corner.
2. Click OK.
Genie information for the selected project displays. For descriptions of the fields on this page, see *Review Genie Configurations*.
3. Choose the type of repair you want to perform:

Repair option alone (Upgrade Project not checked) attempts to fix errors and warnings in a project (used for PowerSCADA Expert 8.0).

Repair option with Upgrade Project checked is used to upgrade projects from previous versions of the product. This option renumbers all genies in the project. Do not perform this option on a project more than once, and do not perform it on PowerSCADA Expert 8.0 projects.

NOTE: When two busbars have the same line active, they are assigned the same busbar number.

The following table describes the repairs made in each option.

Genie Type	Repair	Repair— Upgrade Project
Breaker	Breaker Number	ALL Breaker Numbers
Sources	Source Number Line Active	ALL Source Numbers Line Active
Meters	Meter Number Line Active	ALL Meter Numbers Line Active
Transformers	Sim Source Numbers (top and bottom)	Sim Source Numbers (top and bottom)
Busbars	----	All valid busbars will be reassigned, including destination and source busbars for breakers and transformers

4. Click Repair.
You see a message that describes the degree of repair that is about to take place.
Each message says that graphics pages "will not be modified by the repair process." This means that the repairs will not be applied to your project graphics pages until you press Save.
5. Click Yes to initiate the repair option that you have selected.
A Repair Summary window displays, listing the repairs that have been initiated.
6. To save a .csv copy of this summary, click Export. At the Save As window, type a file name and choose the location at which you want to save the file.
7. Click OK.
The genie information changes, indicating that the repairs have been made.
8. Click Save.
The Save window appears. This is where the changes are saved to your project.
9. Click Yes to save the changes to the graphics pages of the project.
Or click No to back out of making the changes.
If you click No, click Close, then click No when you are asked whether you want to save the modified project.
If you click Yes, the changes are saved to the project. For a large project, this might take several minutes. When the repairs are saved to the project, you see a Save Summary window, which lists the repairs that were made and saved.
Click Export to save a .csv file of these changes. Click OK to exit the summary window and return to the One Line Configuration Utility window.
10. It is possible that some errors and warnings will not be repaired, for example, missing busbar numbers or missing equipment. Click individual errors or warnings to view them (note that the warning and error icons include a tooltip to tell you what is wrong). Note the missing information, then go to the graphics builder to make the necessary changes.
11. Compile the project and then run it.

Automatic Transfer Switch (ATS) Information

ATS Information

The most commonly used information about the ATS genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected ATS.

Basic ATS information includes:

Column	Description
State	Normal (✓), Warnings (⚠), or Errors (✖). See the following table for explanations of errors.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the breaker number for the left side, assigned when adding it to a page of a one-line.
ID2	This is the breaker number for the right side, assigned when adding it to a page of a one-line.
Source Busbar1	The number of the source busbar for the left side.
Source Busbar2	The number of the source busbar for the right side.
Dest. Busbar	The number of the destination busbar.

ATS Errors

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Errors that you might see for ATSs are:

State	Solution
Errors (✖)	
Breaker numbers must be a number greater than 0 and unique.	The breaker numbers are missing, or they are less than or equal to 0. Add or change the breaker numbers.
Source busbar numbers must be a number greater than 0.	The source busbar numbers are missing, or they are less than or equal to 0. Add or change the source busbar numbers.
Destination busbar number must be a number greater than 0.	The destination busbar number is missing, or it is less than or equal to 0. Add or change the destination busbar number.
Source and Destination busbars must not be equal.	The source and destination busbars have the same number; change one number.
Either the Source or Destination Busbar number must exist (busbar may link to a Busbar, transformer, meter, source, or another breaker)	At least one busbar must be linked to this ATS.

Busbar Information

The most commonly used information about the busbar genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected busbar(s).

Basic busbar information includes:

Column	Description
State	Normal (✓), Warnings (⚠), or Errors (✖). See the following table for explanations of errors and warnings.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the busbar number, assigned when adding it to a page of a one-line.

Busbar Errors

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Errors that you might see for busbars are:

State	Solution
Errors (✖)	
Busbar number must be a number greater than 0.	The busbar number is missing, or it is less than or equal to 0. Add or change the busbar number.

Breaker and Switch Information

The most commonly used information about the breaker genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected breaker(s).

Basic breaker information includes:

Column	Description
State	Normal (✓), Warnings (⚠), or Errors (✖). See the following table for explanations of errors.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the breaker number, assigned when adding it to a page of a one-line.
Equipment	The equipment name entered when adding the genie via the Profile Wizard or Automation Interface.
Source Busbar	The number of the source busbar.
Dest. Busbar	The number of the destination busbar.

Breaker and Switch Errors

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Errors that you might see for breakers are:

State	Solution
Errors (✖)	
Breaker number must be a number greater than 0 and unique.	The breaker number is missing, or it is less than or equal to 0. Add or change the breaker number.
Source busbar number must be a number greater than 0.	The source busbar number is missing, or it is less than or equal to 0. Add or change the source busbar number.
Destination busbar number must be a number greater than 0.	The destination busbar number is missing, or it is less than or equal to 0. Add or change the destination busbar number.
Source and Destination busbars must not be equal.	The source and destination busbars have the same number; change one number.
Equipment must be present.	There is no equipment attached to the breaker. Add the appropriate equipment.
Either the Source or Destination Busbar number must exist (busbar may link to a Busbar, transformer, meter, source, or another breaker)	At least one busbar must be linked to this breaker.

Meter Information

The most commonly used information about the meter genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected breaker(s).

Basic meter information includes:

Column	Description
State	Normal (✔), Warnings (⚠), or Errors (✖). See the following table for explanations of errors and warnings.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the meter number, assigned when adding it to a page of a one-line.
Equipment	The equipment name entered when adding the genie via the Profile Wizard or Automation Interface.
Source Busbar	The number of the incoming busbar.
Line Active	The Cicode expression (such as MyTag1 > 0) that determines when the meter detects power on the busbar.

Meter Errors and Warnings

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Warnings indicate settings that might be incorrect. Verify that the settings indicated by the warnings are what you want.

Errors and warnings that you might see for meters are:

State	Solution
Errors (✖)	
Meter number must be a number greater than 0 and unique.	The meter number is missing, or it is less than or equal to 0. Add or change the meter number.
Busbar number must be a number greater than 0.	The busbar number is missing, or it is less than or equal to 0. Add or change the busbar number.
Equipment must be present.	There is no equipment attached to the meter. Add the appropriate equipment.
Busbar number must exist (busbar may link to a Busbar, transformer, meter, source, or breaker)	At least one busbar must be linked to this meter.
Warnings (⚠)	
Line Active should be present.	Line Active should be entered to determine when the meter detects power.
Busbars across all meters should be unique.	Verify that all busbars connected to this meter have the correct, unique, numbers.

Source Information

The most commonly used information about the source genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected source(s).

Basic source information includes:

Column	Description
State	Normal (✔), Warnings (⚠), or Errors (✖). See the following table for explanations of errors and warnings.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the meter number, assigned when adding it to a page of a one-line.
Busbar	The number of the source that powers the connected busbar.
Line Active	The Cicode expression (such as MyTag1 > 0) that determines when the source detects power on the busbar.

Source Errors and Warnings

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Warnings indicate settings that might be incorrect. Verify that the settings indicated by the warnings are what you want.

Errors and warnings that you might see for sources are:

State	Solution
Errors (✖)	
Source number must be a number greater than 0 and unique.	The source number is missing, or it is less than or equal to 0. Add or change the source number.
Busbar number must be a number greater than 0 and unique across sources.	The busbar number is missing, or it is less than or equal to 0. Add or change the busbar number.
Busbar number must exist (busbar may link to a Busbar, transformer, or breaker)	At least one busbar must be linked to this source.
Warnings (⚠)	
Line Active should be present.	Line Active should be entered so the source can detect power on the busbar.

Transformer Information

The most commonly used information about the transformer genie displays by default.

When the Advanced Properties box is checked, the table expands to include everything that is known about the selected transformer(s).

Basic transformer information includes:

Column	Description
State	Normal (✔), Warnings (⚠), or Errors (✖). See the following table for explanations of errors.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the breaker number, assigned when adding it to a page of a one-line.
Source Busbar	The number of the source busbar.
Dest. Busbar	The number of the destination busbar.
Sim. Source	This is the top source number used when adding the transformer.
Sim. Source 2	This is the bottom source number used when adding the transformer.

Transformer Errors

Before you use the drawing, you must correct all errors. Otherwise the project might not compile; and the animation will not work.

Errors that you might see for transformers are:

State	Solution
Errors (✖)	

State	Solution
Source busbar number must be a number greater than 0.	The source busbar number is missing, or it is less than or equal to 0. Add or change the source busbar number.
Destination busbar number must be a number greater than 0.	The destination busbar number is missing, or it is less than or equal to 0. Add or change the source busbar number.
Source and Destination busbars must not be equal.	The source and destination busbars have the same number; change one number.
Either the Source or Destination Busbar number must exist (busbar may link to a Busbar, transformer, meter, source, or another breaker)	At least one busbar must be linked to this transformer.
If a top or bottom source is identified, it must be greater than 0.	The number for the top or bottom source for this transformer must be greater than zero (for voltage-level transformers) or must be left blank (for pass-through transformers).

Work with Alarms

In this section, you will find these topics:

[Alarms Overview on page 138](#)

[Set Up Alarm Pages on page 139](#)

[Add Setpoints and Delays on page 139](#)

[Set Up an Alarm Based on an Enumeration on page 139](#)

[Change an Alarm Severity on page 140](#)

[Enable Waveforms for Onboard Alarms on page 140](#)

[Set Parameters for Event Log Length and Historical Logging of Events on page 140](#)

[Add an Onboard Alarm Tag on page 141](#)

[Set Up Audible Alarms on page 142](#)

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

1. PC-based alarm tags are added in the Profile Editor, when adding each device profile. See [Add Edit or Delete Device Profile on page 41](#) for instructions. For instructions on entering setpoints and delays, see [Add Setpoints and Delays on page 139](#).

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 PowerSCADA Expert.

You cannot configure new onboard alarms from PowerSCADA Expert. You must add the alarm at the device, then you can create the alarm tag for it here. See [Add an Onboard Alarm Tag on page 141](#).

Set Up Alarm Pages

Use this procedure to create separate alarm pages for each alarm type in the project. Follow these steps for each alarm page you want:

1. From Citect Project Editor, click System > Menu Configuration.
2. In the Menu Command line, add the Cicode method that will open the page:

```
PLSDspShowAlarm(INT nType)
```

where *nType* = the type of alarm (e.g., 1=unacknowledged, 3=disabled)

Example (for disabled alarms):

```
PLSDspShowAlarm(3)
```

For information about alarm types, see *AlarmDsp* in the Cicode Programming Reference help file.

Add Setpoints and Delays

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 [Use the Equipment Pop-Up Page on page 164](#) for instructions.

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

```
[Alarm] UseConfigLimits = 1
```

Set Up an Alarm Based on an Enumeration

For instructions on defining an enumeration in the Profile Editor, see [Define an Enumeration on page 66](#).

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 PowerSCADA Expert.
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.

Change an Alarm Severity

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

1. Open the analog alarm in PowerSCADA Expert.
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.

Enable 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 PowerSCADA Expert 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 [Use the Equipment Pop-Up Page on page 164](#) for instructions.

Set 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.

Event Storage: [Alarm]SummaryLength Parameter

The maximum number of alarms that can 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.

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 be used for logging. If you choose to log, you must specify the file format: XML or CSV.

[PLSEventLog]

FileFormat = 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>

Add an Onboard Alarm Tag

When a device onboard alarm has not been included in PowerSCADA Expert, 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.

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 are stored on the device.
3. From the Profile Editor, add the onboard alarm (see [Onboard Alarm Tag Addresses on page 36](#) in Editing Tag Addresses).

Set 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. We support 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

Restore and Back Up Projects

In this section, you will find these topics:

[Restore a Project on page 142](#)

[PowerSCADA Expert Backup on page 143](#)

Restore 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.

PowerSCADA Expert 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 56](#).

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 location that you choose during backup. This is a Citect Zip file; you can open it with WinZip.

Use Cicode to Customize a Project

Cicode is a programming language designed for use in this product 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 before you attempt to use Cicode.

Using Cicode, you can access all real-time data (variables) in the project: variable tags, alarms, trends, reports, and so on. You can also use Cicode to interface with the computer's operating system and communication ports.

For information about driver-specific INI parameters that you can configure, see [Appendix 2: Citect INI Parameters on page 221](#).

For information about other parameters, see the Cicode Reference help file in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

Use Cicode in PLS_Include

Several Cicode modules have been written specifically for use in PLS_Include. This section describes each module.

Click to view descriptions/instructions for each module:

[PLSProviderEngine.ci Module on page 143](#)

[Clear Cache and Refresh Platform on page 145](#)

PLSProviderEngine.ci Module

Use this module when you want to invoke a provider to produce results that can be displayed or acted on in a custom table or report that you create. Providers invoked by this method must be written so that they take a single string as input and return a single string as output.

Module Construction

The following string functions are included in this module:

CallProvider

This function invokes a provider (whose GUID-based identifier must appear in the sProvider argument) with a single string as input (the sArgs argument). The input string

can consist of anything that is meaningful to the provider that you invoke.

The provider then returns a string-based token.

Construction of CallProvider:

```
STRING FUNCTION CallProvider(STRING sProvider, STRING sArgs)

    INT hHandle;
    STRING sResult;
    ErrSet(1);
    sProvider = "^" + sProvider + "^";
    sArgs = "^" + sArgs + "^";
    hHandle = DLLOpen("ProviderGatewayUnmanaged.dll", "MakeRequest",
        "CCC");
    sResult = DLLCall(hHandle, sProvider + "," + sArgs);
    DLLClose(hHandle);
    IF IsError() THEN RETURN "ERROR"; END
    RETURN sResult;

END
```

GetProviderStatus

This function reports the status of a provider invocation by showing the percentage of its completeness. A provider has completed its work when the status reaches 100 percent,

To retrieve status with this function, pass in a token (obtained previously by calling CallProvider) and examine the number contained in the function's return string (from 0 to 100).

Construction of GetProvider Access:

```
-----
STRING FUNCTION GetProviderStatus(STRING sToken)

    INT hHandle;
    INT iPercent;
    ErrSet(1);
    sToken = "^" + sToken + "^";
    hHandle = DLLOpen("ProviderGatewayUnmanaged.dll", "GetPercent",
        "JC");
    iPercent = DLLCall(hHandle, sToken);
    DLLClose(hHandle);
    IF IsError() THEN RETURN "ERROR"; END
    RETURN iPercent;

END
```

GetProviderResult

This function retrieves the result from a provider. Pass a unique token (obtained previously by calling CallProvider) to this function. It returns the provider result as a string. Note that you should only call this function after you verify that the provider work is 100 percent complete.

Construction of GetProviderResult:

```
-----
STRING FUNCTION GetProviderResult (STRING sToken)

    INT hHandle;
    STRING sResult;
    ErrSet (1);
    sToken = "^" + sToken + "^";
    hHandle = DLLOpen("ProviderGatewayUnmanaged.dll", "GetResult",
"CC");
    sResult = DLLCall(hHandle, sToken);
    DLLClose(hHandle);
    IF IsError() THEN RETURN "ERROR"; END
    RETURN sResult;

END
-----
```

Clear Cache and Refresh Platform

When you add, delete, or update a device or topic, you need to shut down and then restart, the PowerSCADA Expert run mode. At that time, we recommend that you also clear the cache and then refresh the platform. This will ensure that data is made available to App Mods.

When you clear the cache, you remove data that is potentially out-of-date. Thus, we recommend that you clear the cache of outdated material.

After you clear the cache, you need to refresh the platform. This step will refresh the Schneider Electric CoreServiceHost, updating its list of devices and topics

Each of these methods uses the PLSProviders.ci module. See [PLSProviderEngine.ci Module on page 143](#) for instructions on creating the statements needed.

PLS_ClearCache

In the Schneider Electric CoreServiceHost, when you call a provider and it returns its result, it caches that result for a given amount of time (which varies by provider). If someone calls that provider again, the system will return the cached result.

If someone adds a device during this time, and then restarts run mode, the device is not available for features like LiveView or basic reporting. Thus, if someone tries to view a table or run a basic report, using the new device, it will not display. The next call that is made to the cache will refresh it.

NOTE: You may also wish to create a graphics page that includes a button that calls the cache and/or refresh.

To clear the cache, call the `PLS_ClearCache` function by doing one of the following:

- If the Schneider Electric CoreServiceHost is on the machine from which you are invoking the function, you can call it with no input parameters:

```
PLS_ClearCache();
```

This may be done during startup or using a button handler.

- If the Schneider Electric CoreServiceHost is on a different machine, you must supply parameters to identify where the Application Services core resides. For example, if the customer's Schneider Electric CoreServiceHost resides on an IO Server named "IO Server1" on "Cluster1", to call `PLS_ClearCache`, enter:

```
PLS_ClearCache("IO Server", "IO Server1", "Cluster1");
```

This cannot be done at startup; you must do it after the startup routine is run. For example, you might use a button handler.

PLS_PlatformRefresh

After you clear the cache, run the platform refresh to update the Schneider Electric CoreServiceHost, causing it to refresh its list of devices and topics.

To run the refresh, call the `PLS_PlatformRefresh` function by doing one of the following:

- If the Schneider Electric CoreServiceHost is on the machine from which you are invoking the function, you can call it with no input parameters:

```
PLS_PlatformRefresh();
```

- If the Schneider Electric CoreServiceHost is on a different machine, you must supply parameters to identify where the Application Services core resides. For example, if the customer's Schneider Electric CoreServiceHost resides on an IO Server named "IO Server1" on "Cluster1", to call `PLS_PlatformRefresh`, enter:

```
PLS_PlatformRefresh("IO Server", "IO Server1", "Cluster1");
```

Customize Default Behaviours

In this section, you will find these topics:

[Time Zone Settings on page 146](#)

[Time Synchronization on page 147](#)

[Trend Tag Scan Intervals on page 147](#)

[Disk Storage Calculation for Trends on page 148](#)

[Deadbands and Ignored Devices and Topics on page 148](#)

To learn about the parameters that you can use to configure drivers, see [Appendix 2: Citect INI Parameters on page 221](#).

Time Zone Settings

The system allows you to interact with devices located in a different time zone than the PowerSCADA Expert 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 device. In addition to the manual procedure, this process can be scheduled to occur periodically (using PowerSCADA Expert 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]\Program Data\ Schneider
Electric\PowerSCADA Expert\v8.0\Applications\Profile Editor
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"

Disk Storage Calculation 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 (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

Deadbands and Ignored Devices and Topics

These two features allow you to limit information that you see in system queries and data acquisition. You set the limits for these features in the *Configuration.xml* file (C:\Program Files (x86) > Schneider Electric > PowerSCADA Expert > v8.0 > Applications > AppServices > bin > Configuration.xml).

Deadbands

```
<ConfigurationItem Key="Deadbands" Category="Platform Mapping"
Application="CitectPlatform">
    <Value />
</ConfigurationItem>
```

Use this line in *Configuration.xml* to reduce the sensitivity to minor changes in real-time data. You can set default deadbands for variable tags. To set a deadband, enter the following in the value field:

```
<Value>XX|NN;</Value>
```

where XX is the IEC 61850 tag name and NN is the percentage. For example, to set Current A to 5% and Current B to 10%, you would enter the following:

```
<Value>mmxu1\A\phsA|5.0;mmxu1\A\phsB|10.0;</Value>
```

Ignored Devices/Ignored Topics

Use these two lines in the *Configuration.xml* to develop a list of devices and topics that you want to ignore in system queries/data acquisition. Typically, you will use this to exclude devices such as the memory device zOL. Ignored devices and topics will not appear in Reporting or LiveView.

To set a value for ignored devices, type the Citect device names (semi-colon delimited) that you want to ignore. For example, to exclude zOL (the one-line memory device) and the network tags device (for monitoring comms loss), type:

```
<Value>zOL;NetworkTagsDev</Value>
```

In the Ignored Topics list, type the topic names (semi-colon delimited) that you want to ignore. Do not include the device name prefix that displays in the Citect project tag names. For example, to exclude AlarmUnhandled and AlarmInvalidTimestamp, type:

```
<Value>AlarmUnhandled;AlarmInvalidTimestamp</Value>
```

Save your changes.

Waveform Management

This chapter discusses how waveforms are stored and associated with alarms. In this section, you will find these topics:

[Waveform Storage on page 150](#)

[Waveform Database and Special Waveform Tags on page 150](#)

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 PowerSCADA Expert 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 I/O server.

Waveform Database and Special Waveform Tags

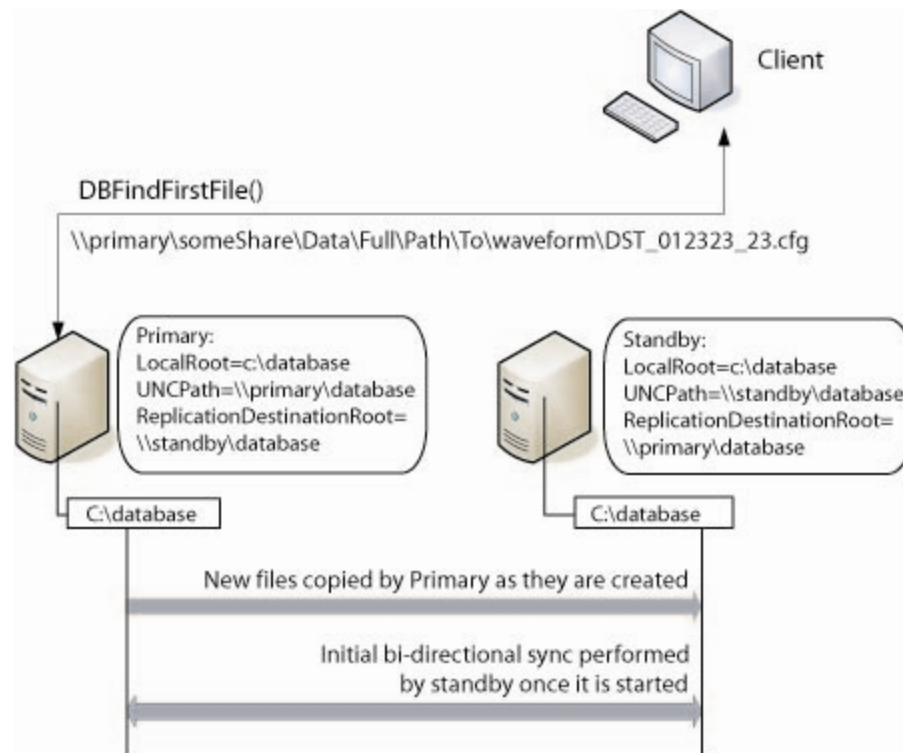
PowerSCADA Expert allows you to browse the waveform database for specific I/O devices. Search for all waveforms within certain time frame is also supported, allowing you to search for all waveforms that could be linked with a given alarm. When you

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.



The Runtime Environment

This chapter discusses the Graphics page in the runtime environment. In this section, you will find these topics:

[*Launch and Close the Runtime Environment on page 153*](#)

[*View the Graphics Page in the Runtime Environment on page 153*](#)

[*Log On to the Runtime Environment on page 153*](#)

[*View the Interface on page 153*](#)

[*View the Alarms/Events Page on page 155*](#)

[*When Alarms do not Display Correctly on page 158*](#)

[*Event/Alarm Log Columns Table on page 159*](#)

[*Alarm/Event Filter Form on page 160*](#)

[*Use the Analysis Page on page 163*](#)

[*Use the Equipment Pop-Up Page on page 164*](#)

[*Perform IEC 61850 Advanced Control on page 167*](#)

[*EcoStruxure WebServices \(EWS\) on page 169*](#)

[*Use Basic Reports on page 169*](#)

[*Set up the Runtime Environment for Basic Reports on page 170*](#)

[*Set Up a Display Client for Basic Report Viewing on page 171*](#)

[*Tasks in the Basic Reporting Application on page 171*](#)

[*Single Device Usage Reports on page 178*](#)

[*Multi Device Usage Reports on page 179*](#)

[*Tabular Reports on page 180*](#)

[*Trend Reports on page 180*](#)

[*Enable Windows Authentication for Basic Reporting on page 181*](#)

[*URL Routing for Basic Reports on page 182*](#)

[*Localization for Basic Reports on page 182*](#)

[*Use LiveView on page 183*](#)

[*LiveView Viewer on page 189*](#)

[*Set Up LiveView on page 192*](#)

[*LiveView Formulas on page 196*](#)

[*LiveView Placeholders on page 195*](#)

[*LiveView Thresholds on page 197*](#)

[*LiveView Formatting on page 194*](#)

[*Multi-Monitor Support on page 202*](#)

[*The Demo Mode on page 202*](#)

[*Application Configuration Utility on page 203*](#)

[*Application Services on page 204*](#)

[*Application Services Host—Citect Data Platform on page 205*](#)

[*Applications on page 206*](#)

[*Single Sign On on page 208*](#)

Launch and Close the Runtime Environment

Launch Runtime: To launch the runtime environment, don one of the following:

- Click Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 >PowerSCADA Expert Runtime.
- From any of the Citect windows, click File > Run.
- From the toolbar of any of the Citect windows, click Run Project:



Close Runtime: There are two ways to shut down the runtime environment:

- Click the “X” in the upper right corner of the screen. Then 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 Manager:



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.

View 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 [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 115.

In this section, you will find these topics:

[Log On to the Runtime Environment](#) on page 153

[View the Interface](#) on page 153

[View the Alarms/Events Page](#) on page 155

[Use Security Viewer](#) on page 161

[Security Viewer Filter](#) on page 162

[When Alarms do not Display Correctly](#) on page 158

[Event/Alarm Log Columns Table](#) on page 159

[Alarm/Event Filter Form](#) on page 160

[Use the Analysis Page](#) on page 163

[Use the Equipment Pop-Up Page](#) on page 164

[Perform IEC 61850 Advanced Control](#) on page 167

Log On to the Runtime Environment

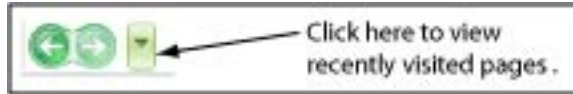
In the upper right-hand corner, click the Login link. Then enter your User ID and password. The features that are available will vary, depending on your user level.

View 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: [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\) on page 115](#). 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 as expected, it could be that a custom genie is not set up correctly. See [Create New Genies on page 116](#) for details on customizing genies.

Lockout/Tagout Icon

This feature causes an icon (see below) to display when a tag attribute for a device indicates that the equipment should be locked out and tagged out. For example, you could assign a tag value from a PLC or other indicating device that reads the condition of the device.

Lockout/tagout icon: 



! DANGER

HAZARD OF ELECTRIC SHOCK, EXPLOSION, OR ARC FLASH

- Do not rely solely on the display of the icon on the one-line. Before working on the equipment, verify that the device is physically locked out/tagged out before performing work on it or any downstream equipment.
- Ensure that all safety regulations and procedures have been followed before you work on the equipment.

Failure to follow these instructions will result in death or serious injury.

See [Enable Lockout/Tagout on page 114](#) for instructions on enabling the lockout/tagout feature.

NOTE: Do not incorrectly configure the tag, as this can lead to unexpected equipment operation. Also consider the possibility of communications loss that could yield false readings.

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.

View the Alarms/Events Page

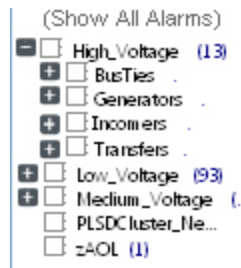
See also: [Use the Equipment Pop-Up Page on page 164](#)

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.

See The Alarm Log, below, for descriptions of color codes.

Equipment Column

On the left side of the page, there is an equipment column. To hide or display this column, click the splitter:



All of the equipment in the project is listed. Most of the equipment is grouped by voltage level. By default, none of the names are checked, which means that information for all of them will display. To list alarms and events for a shortened list of equipment, check the box(es) to the left of the equipment name(s).

The number to the right of the equipment name is the number of active alarms for that equipment.

Filter Information

To filter the information that displays, click Filter, just above the Date column (see [Alarm/Event Filter Form on page 160](#) for more information). From the Alarm Filter window, you can select from a variety of filters.

Remove/Insert/Move Columns

To **remove** a column from the list, right-click its header and select Remove Column.

To **insert** a column, right-click a column header, select Insert Column, then check the name of the column from the dropdown list. The new column displays to the left of the column you right-clicked. If you right-click the white area to the right of existing columns, you will insert the column to the right of the last column.

To **move** a column, left-click the column that is to be moved. Drag the column to the position you want it.

Sort by Column

To sort on the information in a single column (such as the Equipment column), double-click the column header. It will toggle between ascending and descending order.

The Event Log

The Event Log lists alarm/event activity, most recent first (provides sequence of events information). The time is reported to the millisecond. You can display the Message column to see the most detail (such as, "Alarms disabled" and "Alarm xxx acknowledged").

The Alarm Log

To filter the alarms that display, click Filter, just above the Date column (see instructions on using the filter option in [Alarm/Event Filter Form on page 160](#)). From the Advanced Alarm Filter window, you can filter by date range, by text matches for various attributes, or by alarm type.

Notice the alarm colors:

- Acknowledged active alarms display in a **normal red font**.
- Unacknowledged active alarms display in a **bold red font**.
- Acknowledged inactive alarms display in a **normal gray font**.
- Unacknowledged inactive alarms display in a **bold gray font**.

Each alarm provides additional options. To view these options, right-click the alarm. Then you can do the following. Note that these changes will remain only until you leave the page. To set the order, use the parameters,

- 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 the [equipment name Waveform] 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 [Enable Waveforms on page 39](#)).

When the waveform is available for viewing, the Search Waveform dialog displays. From this dialog, click Time Range, and then select the appropriate times; or click All Available to see all waveforms for this equipment. Click OK to display a list of waveforms that fit the date criteria. Highlight the waveform and click View.

After the selected waveform displays, you can view a PDF file that describes the operation of the waveform viewer. Access this file (WaveWeb.pdf in the Citect Bin folder (64-bit example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.0\Bin).

Waveforms must be correctly set up before they will display. See [Enable Waveforms for Onboard Alarms on page 140](#) for more information. See [Use the Equipment Pop-Up Page on page 164](#) 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:

- the alarm has been acquired at the device, but it has not yet been passed to PowerSCADA Expert
- 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

Acknowledge, Silence and Print

Each of the logs includes these buttons:

Button	Description
	Acknowledge Current Page of Alarms: Click to acknowledge all of the alarms that display on the current page.

Button	Description
	Note: You can acknowledge individual alarms in this way: Right-click the alarm that you want to acknowledge, then choose Acknowledge. On a touch screen, tap twice on the alarm row 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: Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.0\Data Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 8.0\Data Notes: When printing: To avoid truncation of data, choose the Landscape orientation. 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).

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: 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. 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. 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. 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: Set alarm filtering in the alarm viewer to reduce the number of alarms that can display. Only support a one-cluster system. If a multiple-cluster system is necessary, display a separate alarm page for each cluster.

Issue	Cause	Resolution
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 Alarm/Event Filter Form on page 160 .
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 Citect column headings that are available for use when viewing the event log and alarm logs. To add a column to the table, right-click the column-heading row, then select Insert Column and choose the column from the list. The column displays to the left of where you right-clicked. To move columns left or right, drag and drop them. To insert a column to the right of the table, right-click the white space next to the existing columns.

To remove a column from the table, right-click its header and select Remove Column.

Column Name	Description
AlmComment	alarm log only: these entries come from the time-stamped digital alarm window "Comment" fields
Area	area, value needs to be set between 0 and 255
Category	event; or high, medium, or low alarm
Change	alarm logs only: when the alarm changes state: first state, second state
Cluster	cluster name to which the alarm belongs
Comment	alarm log only: displays comments from the alarm
Custom3 through 8	custom filters
Date	date (MMDDYYYY) that the event occurred or that the alarm annunciated
Description	description of alarm e.g., Sag Vcn or Under Voltage B-C
Equipment	default equipment name displays; used for alarm filtering and viewing
Help	help page
Location	Onboard or PC-Based
LogState	alarm logs only: The last state that the alarm passed through.
Millisec	alarm logs only: time (MS) that alarm annunciated
Operator	user name from the Citect users list
Priority	the alarm category's priority
Priv	privilege = security level
State	event log: state of the entry in the event log.. event alarm log: disappearance, appearance
Tag	alarm tag
Time	time (HH:MM:SS:MS) that alarm annunciated
Time Quality	This column displays the quality (accuracy) of the time stamp for alarms/events.

Column Name	Description
	Use the "Time Sync" filter to display only data that has confirmed time quality in the log (see Alarm/Event Filter Form on page 160 for instructions on enabling the filter). When there is no SER data, this column reads "No Time Sync Information." When the filter is set to Yes, the view displays only the available time sync information from SER devices. Note: If there is no SER data from any device and the filter is set to Yes, the entire log will be blank.

Alarm/Event Filter Form

This topic describes the PLSCADA filter form. The information in the Citect filter form is the same, but is presented differently on the page. To change the filter form, use the UsePLSFilter parameter:

[Alarm] UsePLSFilter

default: 1 (use PLSCADA filter form)

Change to 0 to use the Citect filter form.

To filter for the information that displays in the alarm logs and 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. Choosing only a start date displays alarms from that date to the current date. Choosing only an end date displays alarms for the past year up to that date. For example, to display alarms only for today's date, enter only a start date.
Start Time/End Time	a time range. 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.). 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 <i>MMXU1\A\phsA</i>, not <i>MainCM4\MMXU1\A\phsA</i>. 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 Description	Alarm Desc from Time Stamped Digital Alarms: a customized on and off text description, such as "active" and "inactive"
Custom Filter	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. 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:	

Filter Option	Description: Display all alarms for:
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 topics; see PowerSCADA Expert help for more information.	
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. See the following table (Categories and Priorities) for a list of the categories and their defaults. 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. As with the category, priority has defaults (see Categories and Priorities table below). You can change these settings in the equipment profiles. <i>However, be sure that you use priority 1 for events.</i>
Time Sync	Yes = in the Alarm or Event Log, only events/alarms with time quality information will be listed. The time sync data displays in the Time Quality column of the log. Data displays to the accuracy recorded at the device. Default: no

Category Label	Category Number	Priority Number
_PLSALM_HIGH	1001	1
_PLSALM_MEDIUM	1002	2
_PLSALM_LOW	1003	3
_PLSALM_EVENT	1004	0

Use Security Viewer

Use this feature to view user activity within your system. This screen lists all user actions that are captured in the Event Log.

View this information from the runtime environment: Click the Alarms/Events tab; then select the Security Viewer subtab.

The screen displays a table with these default columns:

Date	The date that the activity was logged
Operator	User name from the Citect users
Time	The time that the activity was logged
Classification	The class of the event.
Message	From the Message field in the Alarm Log
UserLocation	URL of the computer at which the activity occurred

For more information on these fields, see **Alarm SOE fields** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

To change the view of the log, you can use any of the sort or filter features that are available in the Event Log.

There are three ways to filter information:

- To the left of the log, you can check one or more of the devices in the system. This filters information to include data only for those devices. When nothing is checked, all devices are included.
- You can turn columns "on" and "off." To add columns, right click in the header area of the log, then choose Insert Column. From the list that drops down, you can check an additional column title. The new column displays to the left of the column you clicked.

To remove a column, right click on the header of the column you want to delete, then click Remove Column.
- You can filter that data that is included. To do this, use the Security Viewer filter. For instructions on filtering the columns in the log, see [Security Viewer Filter on page 162](#).

Security Viewer Filter

To filter for the information that displays in the security viewer log, click Filter (in the upper left corner of the screen). The Security View Filter screen displays. The table below describes its settings.

Filter Option	Description: Display all alarms for:
Basic Filter box:	
Start Date/End Date	Choosing only a start date displays alarms from that date to the current date. Choosing only an end date displays alarms for the past year up to that date. For example, to display alarms only for today's date, enter only a start date.
Start Time/End Time	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.). Choosing only an ending time from the start of the day (00:00:00 or 12:00:00 a.m.) through the time selected.
Cluster	This is a single cluster, which was added when setting up the project (listed in Project Editor > Servers > Clusters)
Area	Area (Between 0 and 255). See Alarm SOE fields in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).
Classification	The class of the event. See Alarm SOE fields in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).
Operator	The user ID of the person who has logged on PowerSCADA Expert.
Message	This comes from the Message field in the Alarm Log.
Custom Filter	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. 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.

Use 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 for more information).

Trend data is automatically logged when you check “Trend Tag” for tag and then add it to the 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 (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

- 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:

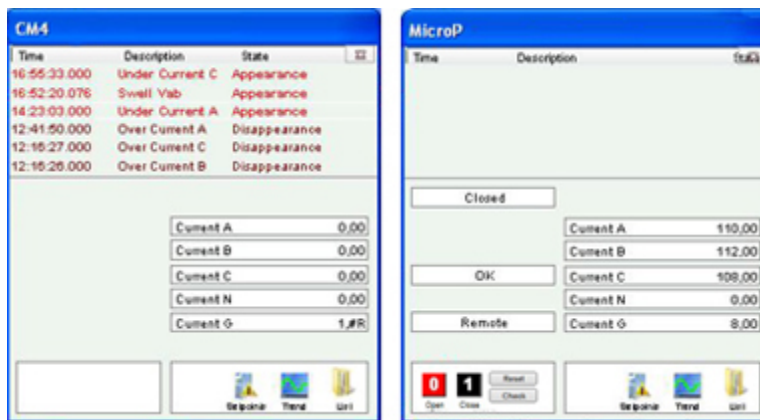


Use the Equipment Pop-Up Page

See also: [View the Alarms/Events Page on page 155](#)

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 [Add and Modify User Accounts on page 210](#) 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, and so on). 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 [Enable Waveforms on page 39](#)).

When the waveform is available for viewing, it displays when you click this link. For information about how the waveform viewer works, see the WaveWeb.pdf file in the Citect bin folder (64-bit example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\v8.0\bin).

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 [Enable Waveforms for Onboard Alarms on page 140](#) for more information. See [Use the Equipment Pop-Up Page on page 164](#) 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 PowerSCADA Expert
- 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)

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\AphaA 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.

Perform IEC61850 Advanced Control

To begin using the advanced control feature, click Check in the lower left section of the window. See [Set Up IEC 61850 Advanced Control on page 117](#) for information on setup. See [Perform IEC 61850 Advanced Control on page 167](#) for information on performing this advanced control.

View Waveforms

After you select a waveform for viewing from the genie status page, the external waveform viewer displays it. For instructions on using the tool's analysis feature, see WaveWeb.PDF, located in the Bin folder of the PowerSCADA Expert 8.0 Bin folder (example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.0\Bin).

Enter 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.



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.

View 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.

View 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 [Add and Modify User Accounts on page 210](#). 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 Override Tag Status**, below, for details.

Override 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.

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 information, see the **I/O Device Properties** topic in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

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 colors are assigned for the tag override and control inhibit. Change the default background colors in the parameters, not in the ini file.

To change the color for tag overrides, use OverrideTextBackgroundColor. To change the color for control inhibits, use ControlInhibitTextBackgroundColor. For detailed help, see Page Parameters in the Parameters.chm help file (Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 > PowerSCADA Expert Web-based Help).

Perform IEC 61850 Advanced Control

The advanced control window provides these options for IEC 61850 IEDs:

- run synchro check on the selected equipment
- run interlock check on the selected equipment
- send a command to open or close the equipment

You can either check the features without sending an open/close command, or you can send an open/close command without running the checks..

Only users who have privilege level of Engineer or Admin can perform these checks or operate the equipment.



WARNING

UNINTENDED EQUIPMENT OPERATION

- Do not incorrectly configure the tag, as this can lead to unexpected equipment operation.
- Ensure that you understand the effects of using the "bypass" option so you do not shut down critical equipment.
- Also consider the possibility of communications loss that could yield false readings.
- Ensure that all safety regulations and procedures have been followed before you work on the equipment.

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

NOTE: For information on enabling this feature, see *Set Up IEC 61850 Advanced Control* on page 117

To access the advanced control window, open the equipment genie pop-up page on the one-line. Note that, after you enable this feature, there is a "Check" button on the lower left:

Time	Description	State
10:23:56.663 AM	Communication Failure	Disappearance

Closed	
Racked In	Current A 105.80
Healthy	Current B 96.90
OK	Current C 133.40
Comm Ok	Current N 0.00
Remote	

0

Open

1

Close

Reset

Check

Setpoints

Trend

List

Click Check, and the advanced control window opens:

CSW11Pos

Synchro Check	Check	Bypass
Interlock Check	Check	Bypass
Test	Operate	Test
Value	Open	Close

Send

Synchro Check: Use synchro check to verify that the waveforms for the equipment's power factor, voltage, and current are all aligned.

On the Synchro Check line, click Check to perform the synchro check, or click Bypass to ignore the synchro check. Default: Check.

Interlock Check: Use interlock to verify that there are no blocking conditions that need to be considered before switches are opened or closed.

On the Interlock Check line, click Check to perform the interlock check option, or click Bypass to ignore the interlock check. Default: Check.

Test: Click Operate if you want to send the command to the equipment and to complete the "value" setting. Click Test if you want to send the command to the equipment, and to verify the synchro and/or interlock statuses, but not complete the "value" setting. Default: Operate.

NOTE: If you choose "Check" for the synchro and/or interlock checks and "Operate" for the Test line, the open/close operation will not occur if the equipment fails the checks.

Value: Choose the command that you want to send to the equipment: open or close. Default: Closed if the breaker is open; otherwise, Open.

Send: Click to send the command to the device to perform the action(s) that you selected.

EcoStruxure WebServices (EWS)

EcoStruxure Web Services (EWS) for PowerSCADA Expert was developed to share real-time, historical, and alarm data with StruxureWare Building Operations (SBO) and historical data with Power Monitoring Expert (PME). Do not confuse this feature with the EWS server that was released as a part of PowerSCADA Expert/Vijeo Citect version 7.40 (which is for tag level process data).

EWS uses web-based http protocol to transfer data. It enables two-way data transfers, which allows the acknowledgment of alarms from SBO. To include this new EWS implementation in your installation, select the EWS Server checkbox during installation.

EWS is set up and configured using the Application Configuration Utility (Start > All Programs > Schneider Electric > PowerSCADA Expert 8.0 > Config Tools > Application Config Utility). For details, see [EcoStruxure Web Services Setup on page 204](#).

Use Basic Reports

The reporting feature is an IIS web application that is typically hosted on the same server as the PowerSCADA services. The PLS_Include project defines a *PLS_ReportPage* (along with its screen resolution-specific variant pages). *PLS_ReportPage* contains a Microsoft Web Browser ActiveX control in which the reporting web pages are displayed.

Before you can set up Basic Reports and generate/view reports, you must set up data acquisition parameters. To do this, use the Application Configuration Utility. See [Application Services Host—Citect Data Platform on page 205](#) and (for receiving reports via email) [Basic Reports on page 207](#) for instructions.

In System > Menu Configuration, menu tabs are configured to use the new "PLS_ReportDsp()" Cicode function to send URLs to the web browser control at runtime. The control then browses to the available reporting web pages. See the PLS_Example project for examples of this functionality.

When switching between PowerSCADA Expert projects in runtime, you must restart the Schneider Electric Service Host (CoreServiceHost) service before you run the reporting application. This allows the reporting application to load data from the currently running PowerSCADA Expert project.

To get started setting up a report, see [Set up the Runtime Environment for Basic Reports on page 170](#).

For descriptions of each report type, see [PowerSCADA Expert Basic Reports on page 178](#).

NOTE: If you install Matrikon Explorer on the same computer as PowerSCADA Expert 8.0, the LiveView and reporting features will not launch. To prevent this, install Matrikon before you install PowerSCADA Expert 8.0. If you install Matrikon after you install PowerSCADA Expert 8.0, you need fix the issue in this way: Go to IIS > ISAPI Filters, and then reset the dll that is already selected (click browse and re-select *v4.0.30319 aspnet_filter.dll*). Click OK.).

Set up the Runtime Environment for Basic Reports

Follow these steps to add new items to the project, add the necessary INI parameters for CtAPI and basic report security, and create the CtAPI connection for reporting.

Create the Menu Items for Report Page

For a complete discussion of reporting web application URLs, see *Create and View Basic Reports* on page 171.

The following steps describe how to interact with the reporting web application via the runtime environment.

1. From Citect Project Editor, click System > Menu Configuration.
2. Add the new menu item that you want for each of your basic reports.
3. In each of these menu items, in the Menu Command line, add the Cicode method that will display a report tab. You can create your own custom method or use the default:

```
PLS_ReportDsp(STRING sIPAddress, STRING sName, STRING sOptions = "", STRING sTitle="")
```

Examples:

```
PLS_ReportDsp("10.10.10.10", "SingleDeviceReport",  
"ShowConfiguration/MyConfiguration", "Single Device Usage Report");
```

which opens a configured single device report with the saved configuration name "MyConfiguration"

or

```
PLS_ReportDsp("10.10.10.10", "SingleDeviceReport", "", "Single Device Usage  
Report");
```

which opens an unconfigured single device usage report at the parameters entry page

Add the Following INI Parameters

To allow trend queries that yield the desired amount of historical data:

```
[Trend]MaxRequestLength = 100000000,  
allowable range: 1–100000000  
(example: a value of 70080 would yield two years of data for one device/one topic,  
assuming 15-minute trends)
```

To allow CtAPI to connect remotely:

```
[CtAPI]Remote = 1
```

To define a privilege level for users to view reports:

```
[Reporting]PrivLevel - Default = 0
```

To define an area for users to view reports:

```
[Reporting]Area - Default = 0
```

Set Up a Display Client for Basic Report Viewing

To be able to properly interact with the basic reporting web application at a display client, you must set a registry key to force the Microsoft Web Browser ActiveX control to use Internet Explorer 9 emulation.

Create the following DWORD value at the following registry key path:

Path: HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Microsoft\Internet Explorer\MAIN\FeatureControl\FEATURE_BROWSER_EMULATION

Value Name: Citect32.exe

Value: 9999 (decimal)

NOTE: This registry setting affects the Citect32.exe process only. It has no effect on other applications that use the MS Web Browser ActiveX control.

Tasks in the Basic Reporting Application

You can use the following tasks within the reporting application to create, view, and email basic reports:

[Perform IEC 61850 Advanced Control on page 167](#)

[Create and View Basic Reports on page 171](#)

[Configure Email Settings to Send Basic Reports on page 173](#)

[Send Basic Reports via Email on page 174](#)

[Read, Export, Print, and Edit Basic Reports on page 176](#)

Create and View Basic Reports

This section explains how you can create basic reports and save report configurations using a web browser (e.g., Internet Explorer).

For instructions on interacting with the reporting web application via the runtime environment, see [Set up the Runtime Environment for Basic Reports on page 170](#).

You can generate basic reports in two ways:

- run a new report by entering parameters
- run a report from a saved configuration

Run a New Basic Report

There are two options for this:

1. Browse directly to the URL of the report. Use this format:

`http://<ServerName>/Reporting/Report/<ReportName>`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

Standard report names are:

- MultiDeviceReport
- SingleDeviceReport
- TabularReport
- TrendReport

2. Browse to the default reporting URL, and click the report you want to view.

An example of this format is:

`http://<ServerName>/Reporting/`

where:

`<ServerName>` = the name or IP of the reporting server

Additional formats are required, depending on the operating systems. For more information, see [URL Routing for Basic Reports on page 182](#).

Run a Basic Report and Save its Configuration

NOTE: If you plan to view a basic report using [Rapid Access Labels on page 200](#), you must save a configuration. After it is saved and you generate a rapid access label, do not change the configuration name. If the configuration name is changed, you must generate a new rapid access label.

To create and save a basic report configuration so you can view it later:

1. Visit the "build configuration" URL of the report you wish to use. Use this format:

`http://<ServerName>/Reporting/Report/<ReportName>/BuildConfiguration`

where:

`<ServerName>` = the name or IP of the reporting server

`<ReportName>` = the name of the report you wish to view

2. Enter the report query parameters.

NOTE: If you enter a fixed date range, all reports that you generate with this configuration will use that date range. The best practice is to use one of the relative date ranges, such as "last month."

3. After the report displays, scroll to the bottom of the page. You should see a text box with XML. This XML represents your saved report configuration. Copy the entire contents of the text box into the text editor of your choice.
4. Save this new file to the Reporting\ReportConfigurations\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

`C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA Expert Reporting\Reporting\ReportConfigurations\`

The filename **must** be in the following format:

`<ReportName>_<ConfigurationName>.cfg`

where:

`<ReportName>` = the name of the report (such as MultiDeviceReport, SingleDeviceReport)

`<ConfigurationName>` = a name for this configuration (alphanumeric only)

If you use Notepad, ensure that you apply the correct suffix (.cfg) and not the default (.txt).

View a Basic Report Using a Saved Configuration

To view a basic report with a saved configuration, browse to the URL of the report and specify the configuration. This method runs the report directly with the saved configuration (you cannot change the parameters).

This URL is in the following format:

`http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

<ReportConfiguration> = the name of the saved configuration to use

Modify and View a Basic Report Using a Saved Configuration

Use this procedure to modify a configuration that you saved according to "Run a Basic Report and Save its Configuration," above.

Browse to the "show configuration" URL for the report that you want to modify. Use this format:

`http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>/ShowConfiguration`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you want to view/modify

<ReportConfiguration> = the name of the saved configuration to use

Run the report as you normally would, editing selections on the parameter entry page as necessary.

After the report runs, a text box displays at the bottom of the finished report. It contains the new XML of your saved configuration.

Copy and paste this new XML into your saved configuration file (overwriting the old XML).

Remove a Saved Configuration

To remove a saved configuration, delete the saved configuration file from Reporting\ReportConfigurations\ directory, described above:

Example (64 bit):

C:\Program Files (x86)\Schneider Electric\PowerSCADA
Expert\v8.0\Applications\Reporting\ReportConfigurations

Configure Email Settings to Send Basic Reports

Follow these steps to configure email settings so that you can send PowerSCADA Expert basic reports to multiple email addresses.

NOTE: You must configure the SMTP server and email list(s) before you email reports. See [Send Basic Reports via Email on page 174](#) for instructions on sending these emails.

SMTP Server and From Address

For instructions on setting up the SMTP server, see [Basic Reports on page 207](#).

Email Lists

Before you can send email via the URL or ReportMailer method, you must create at least one email list:

1. Open your preferred text editor; enter one or more email addresses (one per line, no commas).
2. Save this text file in the Reporting\ReportConfigurations\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA Expert Reporting\Reporting\ReportConfigurations\

The filename must be in the following format:

Email_<EmailListName>.cfg

where:

<EmailListName> = an alphanumeric (no spaces) name for the email list (for example, Administration)

Email Body

The email body that you send is contained in a resource (.resx) file in the Reporting\bin\Resources\Reporting.en-US.resx\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA Expert Reporting\bin\Resources\Reporting.en-US.resx\

The email body is the same for all Report Configurations and Email Lists, but you can modify the entry for ReportEmailBody to change the body of the email that is sent.

Send Basic Reports via Email

Before you can send PowerSCADA Expert basic reports via email, you must configure the SMTP server and email list(s) before you email reports. See [Configure Email Settings to Send Basic Reports on page 173](#) for instructions on configuring this information.

There are three ways to send basic reports via email:

1. The Report Viewer email button

Use this method to send a customized one-time email to an individual or group of email addresses.

1. Run the report as normal.
2. Click the Email icon () in the report viewer.
3. Enter the requested information in the pop-up dialog.
4. Click Send.

2. Visit a Specific URL

NOTE: Each visit to a URL causes the email to be sent. Be sure that you have the correct report and email list before you visit this URL/send the email. Also, you should secure this URL using the web.config file. For information on modifying/using the web.config file, see <http://support.microsoft.com>, and search on kb 815179.

To send a basic report to an existing email list, visit the following URL:

`http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>/Email/<EmailList>`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

<ReportConfiguration> = the name of the saved configuration to use

<EmailList> = the name of the email list you wish to use

You must use a saved configuration (see [Create and View Basic Reports on page 171](#) for instructions). You cannot change report parameters from this URL.

No progress bar or update will display, as these interfere with some scheduling clients.

3. Use Cicode via ReportMailer

You can use a utility called ReportMailer to email basic reports. This command line utility is located in the PLS_Include project. It can be called by Cicode. You can create a button on the graphics page and have it call the Cicode function or use a scheduled process to trigger an email.

Before you can use ReportMailer, you need to create or edit the file called *ReportMailer.ini* file that is in your project (not in PLS_Include). The ReportMailer.ini file must include the text listed in the table below:

Text Field	Required Setting	Description
LoginUsername	demo	Username for logging in to reporting system for emailing reports
LoginPassword	demo	User's password, will be encrypted on the first run
IsEncrypted	False	Flag that indicates if the password is encrypted. If you change the password, edit the field (replacing the unreadable encrypted entry, if one exists). Then change this value to False. The new password will be encrypted at the next startup cycle, and this field will be updated to True.
ScadaBinPath	C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.0\Bin	The bin path of PowerSCADA Expert
LogOnUrl	http://SCADASERVER/Reporting/LogOn.aspx	The URL of the logon page(this is an example; use your own server name)
ReportServerName	SCADASERVER	The name or IP address of the server running the reporting application
LogLevel	All	The level of logging you want in the report mailer application. This log is saved to a ReportMailerLog.txt file in the running project's directory. Possible settings are ALL, DEBUG, ERROR, WARN.

After this file is configured, run the ReportMailer.exe with the following syntax:

`ReportMailer.exe <ReportName> <ReportConfiguration> <EmailList>
<ScadaProjectPath>`

where:

<ReportName> = the name of the report you wish to view
<ReportConfiguration> = the name of the saved configuration to use
<EmailList> = the name of the email list you wish to use
<ScadaProjectPath> = the full path to your SCADA project

This command line application may be called from Cicode following the below example:

FUNCTION

```
PLS_EmailReport()
  ErrSet(1);
  STRING FilePath = ParameterGet("CtEdit","User","") + "PLS_
  Include\ReportMailer.exe " +
  "MultiDeviceReport SampleConfiguration SampleList " +
  "^"C:\ProgramData\Schneider Electric\PowerSCADA Expert
  8.0\User\PLS_Example^"";
  Exec(FilePath);
END
```

NOTES

- The SCADA project path must be enclosed in escaped quotes (^).
- This is an asynchronous (non-blocking) call. While the EXEC() method will return immediately, it may take a few moments to run and email the report. See the web.config timeout value (see option 2 above) for more information.
- You can also call the ReportMailer application directly from a command line. In this case, you can add the term "blocking" to the command line (as a fifth parameter). This causes ReportMailer to act in a synchronous state (block the call) and to return any error messages to the console. Never use the "blocking" parameter by Cicode, as it could prevent EXEC() from returning in a timely fashion.

Scheduling Basic Reports

You can schedule the emailing of basic reports by executing the above Cicode as an action from a timed event. For more information, see **Configuring Events** in the CitectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

You can also use the Windows Task Scheduler to send these reports. Refer to Microsoft's documentation on the task scheduler at: <http://windows.microsoft.com/en-US/windows7/schedule-a-task>.

Read, Export, Print, and Edit Basic Reports

After you create a basic report, you can:

- change its appearance (Page Setup)
- print it
- change its view: HTML or PDF
- scroll to the beginning, previous page, next page, or last page
- export it to a variety of formats
- email it

The toolbar options table below describes these options.

Toolbar Options

The report toolbar includes several options:

Option	Description
Parameters/Report	Toggle between viewing the parameters (setup) page and the report.
Parent/Child reports	Not currently used.
Hide/Show	Not currently used.
Page Setup	Click this link to open the Page setup window, where you can determine paper size, and page orientation and margins.
Print	Click this link to print the report. NOTE: For best formatting of the report, you should export to PDF, and then print.
Print Preview	In HTML mode, click this link to view the print output.
Viewer Mode	You can view in HTML or PDF mode. Select the mode, then click Viewer Mode to change the view.
Viewer Mode Set ()	Click to confirm the choice of viewer mode.
Pagination 	Click the left and right arrows to page backward and forward in the report. Or type the page number you want to see.
Select a format	For exporting, choose the format (not HTML) that you want.
Export ()	See instructions below for exporting a report.
Email ()	Click this link, and then enter the requested information. Click Send. For other ways to email reports, see Send Basic Reports via Email on page 174 .

Export a Basic Report

Before you can print a basic report, you must export it into a format that can be printed.

You can export to:

- Acrobat (Pdf) File
- Web Archive
- Word Document
- Xml File
- XLS Document

To export:

1. While viewing the report, select a format, then click Export ().
2. Type the location at which you want to save the file.
3. Set any other properties you wish.

4. Click Export.

Edit the Basic Report Appearance

With the report displayed, you can:

- Change the paper size
- Change the paper source
- Change the page orientation
- Change the page margins
- Change the number of pages per sheet
- Add a watermark

PowerSCADA Expert Basic Reports

During runtime, you can create, view, save, and print basic reports.

Before you can view basic reports, there must be data logged for the project. See [Use the Analysis Page on page 163](#) for help.

You must also create a report tab for the graphics page in runtime mode. See [Set up the Runtime Environment for Basic Reports on page 170](#).

NOTE: If the Schneider Electric CoreServiceHost has not been refreshed after devices or topics have been added, you should clear the cache and refresh the platform in order to access the new devices or topics.

See [Clear Cache and Refresh Platform on page 145](#) for instructions.

After you have logged trend information, you can create and view basic reports. From the graphics page in the runtime environment, click the reports tab and choose the basic report type you want to create:

[Single Device Usage Reports on page 178](#)

[Multi Device Usage Reports on page 179](#)

[Tabular Reports on page 180](#)

[Trend Reports on page 180](#)

[Rapid Access Labels on page 200](#)

See also:

[Localization for Basic Reports on page 182](#)

Single Device Usage Reports

Use single device usage reports to display historical energy data from a single device and multiple topics. A single device report includes only usage and consumption topics.

NOTE: The report is optimized for up to five topics. If you choose too many topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See [Set up the Runtime Environment for Basic Reports](#).

To set up a single device usage report, follow these steps:

1. Browse to the single device report in the reporting web application. When prompted, enter your Citect user account information. Click Login.
2. At the next screen:

- Type a report title.
 - In Reporting Period, choose the date range for this report, for example, *last week*.
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
 - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)
 - Highlight the name of the device that you want for the report.
 - Check the topics to be included.
3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a usage summary table, and a graph and table for each topic you selected. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

Multi Device Usage Reports

Use multi device usage reports to display historical energy data for multiple devices and one topic. A multi device usage report includes only usage and consumption topics.

NOTE: If you choose too many devices, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a multi device usage report, follow these steps:

1. Browse to the multi device usage report in the reporting web application. When prompted, enter your Citect user account information. Click Login.
2. At the next screen, do the following:
 - Type a report title.
 - In Reporting Period, choose the date range for this report, for example, *last week*.
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
 - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)
 - Check the names of the devices for the report.
 - Highlight the topic to be included.
3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a usage summary, a value table by interval for all of the devices selected, and a pie chart. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

Tabular Reports

Use tabular reports to display a system's historical data in a table format. Tabular reports can include one or more devices and one or more topics. A tabular report can include all available topics.

NOTE: The report is optimized for up to five topics. If you choose too many devices or topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a tabular report, follow these steps:

1. Browse to the tabular report in the reporting web application. When prompted, enter your Citect user account information. Click Login.
2. At the next screen, do the following:
 - Type a report title.
 - In Reporting Period, choose the date range for this report, for example, *last week*.
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
 - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)
If you leave the default *By Interval*, you will get every data point in the selected date range.
 - Check the name(s) of the device(s) for the report.
 - Check the topic(s) to be included.
3. Click Generate Report.

After the report is generated, it displays as a table on the screen. It lists data for all of the tags according to their timestamps. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

Trend Reports

Use trend reports to display a system's historical data in a trend (line) and table formats. Trend reports can include one or more devices and one or more topics. A trend report can include all available topics.

NOTE: The report is optimized for up to five topics. If you choose too many devices or topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a trend report, follow these steps:

1. Browse to the trend report in the reporting web application. When prompted, enter your Citect user account information. Click Login.
2. At the next screen, do the following:
 - Type a report title.
 - In Reporting Period, choose the date range for this report, for example, *last week*.

If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)

- From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)

If you leave the default *By Interval*, you will get every data point in the selected date range.

- Click the name(s) of the device(s) for the report.
- Click the topic(s) to be included.

3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a trend for each topic included (selected data points over the period of the trend) followed by a table with every timestamp in the period selected. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

Enable Windows Authentication for Basic Reporting

You may want to use Windows Authentication for logging in to the Basic Reports application. This could be to authenticate from Active Directory or to provide a single-sign-on. Follow the steps below to configure Windows Authentication.

NOTE: These steps are specific to Windows 7; they may be different for other operating systems. For further assistance, view Microsoft's documentation on this topic at:

[http://technet.microsoft.com/en-us/library/cc754628\(v=ws.10\).aspx](http://technet.microsoft.com/en-us/library/cc754628(v=ws.10).aspx)

1. Turn on the Windows Authentication feature:
 - a. From the control panel, click Programs and Features > Turn Windows features on or off.
 - b. Check Windows Authentication.
2. Enable Windows Authentication in IIS:
 - a. From the Control Panel > Administrative Tools, choose Internet Information Services (IIS).
 - b. Select the root node from the tree on the left (or the Reporting website, if this server hosts multiple sites).
 - c. From the right hand pane, in the IIS section, click Authentication.
 - d. Enable Windows Authentication.
3. Modify web.config to specify Windows Authentication:
 - a. From the root of the Reporting direction, locate web.config.
 - b. Change the line:

```
<authentication mode="Forms">
to
<authentication mode="Windows">
```
4. Add roles to web.config:

You must add roles to web.config to allow access to the basic reporting application. For example, to allow the role (group) Administrators, add the following to the web.config file:

```
<authorization>
```

```
<allow roles="Administrators"/>  
<deny users="?" />  
</authorization>
```

Modifying the web.config file is an advanced topic that is covered on the Microsoft website.

See the following link for instructions provided on the Microsoft website:

<http://www.iis.net/configreference/system.webserver/security/authentication/windowauthentication>

Additional information may be available in the following knowledge base article:

<http://support.microsoft.com>, and search on kb/815179.

URL Routing for Basic Reports

The basic reporting application uses Asp.NET extensionless URL routing. Depending on your operating system, you may need to follow additional steps to enable URL routing in your project.

Windows 2008 R2 and Windows 7

Microsoft has discovered an issue with extensionless URL routing in certain installations of IIS 7.0 and IIS 7.5. To address this issue, Microsoft has released a hotfix referenced by KB article 980368. This hotfix is available at <http://support.microsoft.com/kb/980368>.

This hotfix is included in Service Pack 1 for Windows 2008 R2 and Windows 7. To receive the hotfix, you should install Service Pack 1. This installation provides additional important updates to the operating system. To obtain Service Pack 1 for Windows 2008 R2 and Windows 7, go to either Windows Update or <http://support.microsoft.com/kb/976932>.

Windows XP

When you install IIS on a Windows XP operating system, IIS version 5.1 is installed. When browsing to the default URL, you must append to the URL "default.aspx".

Example:

<http://<servername>/Reporting/default.aspx>

Localization for Basic Reports

To localize basic reports, you must first rename and translate the .resx files into the target language(s), called "culture names." The .resx files are located in the PowerSCADA Expert Resources directory (default location: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\v8.0\Applications\AppServices\bin\Resources).

Make copies of the .resx files listed in the table below. Change the extensions of the copies into the culture name for the target language. Note that you do not need to include both the culture ("es") and the region ("ES"), unless you specifically want a language that is different from the default.

For example, BinningResources.en-US.resx would be named BinningResources.es-ES.resx when you localize it into Castillian Spanish.

For a list of the culture names you can use, see the *National Language Support (NLS) API Reference*, currently at:

[NLS API](#)

After you change the extensions, provide translations for the strings in each .resx file.

Store the localized .resx files in the same Resources folder as the English versions.

Template Editor

Use the Template Editor to begin creating, duplicating, modifying, and deleting LiveView templates and views.

See [Set Up LiveView on page 192](#) for complete instructions.

Use LiveView

PowerSCADA Expert provides the ability to create and view LiveView templates and views. Some basic predefined templates are included with the software; you can create new templates or make copies of the predefined templates and edit the copies.

Before you view LiveView templates and views, you must set up data acquisition parameters. To do this, use the Application Configuration Utility. See [Application Services Host—Citect Data Platform on page 205](#) for instructions.

NOTES:

- If you find that a predefined table does not include enough cells for the data you want to display, use the duplicate feature to make a copy of the predefined table. Then add the needed cells to the duplicate.
- If you install Matrikon Explorer on the same computer as PowerSCADA Expert 8.0, the LiveView and reporting features will not launch. To prevent this, install Matrikon before you install PowerSCADA Expert 8.0. If you install Matrikon after you install PowerSCADA Expert 8.0, you need fix the issue in this way: Go to IIS > ISAPI Filters, and then reset the dll that is already selected (click browse and re-select *v4.0.30319 aspnet_filter.dll*). Click OK.).

You can only view data in these templates if your system is online and you are connected to devices that provide data.

To begin creating LiveView templates, click Start > Programs > Schneider Electric > PowerSCADA Expert 8.0 > Config Tools > Table Editor. The LiveView Template Editor screen displays.

To view a template, see [LiveView Viewer on page 189](#).

To create a new template, see [Set Up LiveView on page 192](#).

See also:

[LiveView Formulas on page 196](#)

[LiveView Placeholders on page 195](#).

[LiveView Thresholds on page 197](#)

[LiveView Formatting on page 194](#)

LiveView Tables

Click any of the following links to learn about the live view tables:

- [LiveView Basic Readings Summary on page 184](#)
- [LiveView Power Flow Summary on page 184](#)
- [LiveView Energy Summary on page 185](#)
- [LiveView Energy Readings on page 185](#)
- [LiveView Fundamental Phasor Readings on page 185](#)
- [LiveView THD Current Summary on page 185](#)
- [LiveView THD Voltage Summary on page 186](#)
- [LiveView Uptime Summary on page 186](#)
- [LiveView Incremental Reactive Energy Summary on page 186](#)
- [LiveView Incremental Real Energy Summary on page 187](#)
- [LiveView Harmonic Apparent Power Flows on page 187](#)
- [LiveView Harmonic Reactive Power Flows on page 188](#)
- [LiveView Harmonic Real Power Flows on page 188](#)
- [LiveView Demand Current Summary on page 189](#)
- [Live View Demand Voltage Summary on page 189](#)

LiveView Basic Readings Summary

This summary displays real-time basic power information for a selected device or devices. After opening the basic readings summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The basic readings summary provides this data:

- voltage A-B (V)
- current A (A)
- real power (kW)
- power factor

LiveView Power Flow Summary

This summary displays a power flow summary for your system devices. Use the information from this table to help optimize the system's power flow.

After opening the power flow summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The power flow summary provides this data:

- real power (kW)
- reactive power (kVAR)
- apparent power (kVA)
- demand average (kW)
- demand peak (kW)
- predicted demand (kW)

LiveView Energy Summary

This summary displays an energy summary for your system devices. Use the information from this table to help monitor the system's energy consumption.

After opening the energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The energy summary provides this data:

- real power (kW)
- reactive power (kVAR)
- apparent power (kVA)
- block demand real power (kW)
- thermal demand real power (kW)
- peak block demand real power (kW)
- peak thermal demand real power (kW)
- block demand real power predicted (kW)
- thermal demand real power predicted (kW)

LiveView Energy Readings

This table displays accumulated energy readings for a single device. Data is accumulated beginning with the last energy reset for the device.

Energy values, will be according to one of these accumulation methods:

Absolute (unsigned): The device stores positive energy values, regardless of the direction of power flow. The energy value increases, even during reverse power flow.

Signed: The device stores both positive and negative energy values. The energy value increases or decreases, depending on the direction of the power flow.

After opening the live view energy readings table, choose the device you want and set the update interval for this table. Click Display Table.

The live view energy readings table provides these accumulated readings:

- real energy (kWhr)
- reactive energy (kVARhr)
- apparent energy (kVAhr)

LiveView Fundamental Phasor Readings

This summary displays a fundamental phasor readings table for a single device, to confirm that the system is properly wired.

After opening the fundamental phasor reading template, choose the device for which you want readings and set the update interval for this table. Click Display Table.

The fundamental phasor readings table provides a phasor diagram that indicates current and voltage magnitudes and angles for each phase.

LiveView THD Current Summary

This summary displays a THD current summary for your system devices. Use the information from this table to monitor your equipment and system power quality.

After opening the THD current summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The THD current summary provides this data:

THD current (%):

- phase A
- phase B
- phase C
- neutral

LiveView THD Voltage Summary

This summary displays a THD voltage summary for your system devices. Use the information from this table to monitor your equipment and system power quality.

After opening the THD voltage summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The THD current summary provides this data:

THD voltage (%):

- Vab
- Vbc
- Vca
- Van
- Vbn
- Vcn

LiveView Uptime Summary

This summary displays an uptime summary for your system devices. Use the information from this table to view the number of seconds the equipment has been in uptime (defined as all three phases > 10% nominal), and to view the percentage of uptime vs. total time. The summary includes the last 12 months.

After opening the uptime summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The uptime summary provides this data:

For the past 12 months:

- Uptime %
- Uptime
- Downtime

LiveView Incremental Reactive Energy Summary

This summary displays an incremental reactive energy summary for your system devices. Use the information from this table to monitor transmission of energy beyond the previous baseline, to help achieve optimum loading.

After opening the incremental reactive energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The incremental reactive energy summary provides this data:

- incremental reactive energy into the load (kVARHr)
- incremental reactive energy out of the load (kVARHr)
- date/time of the last incremental energy update

LiveView Incremental Real Energy Summary

This summary displays an incremental real energy summary for your system devices. Use the information from this table to monitor the energy usage and production above the previous baseline, to help achieve optimum loading.

After opening the incremental real energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The incremental real energy summary provides this data:

- incremental real energy into the load (kVARHr)
- incremental real energy out of the load (kVARHr)
- date/time of the last incremental energy update

LiveView Harmonic Apparent Power Flows

The harmonic apparent power flows table displays real-time apparent power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic apparent power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.

You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic apparent power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- remaining hold
- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
 - power flow in from the utility
 - power flow out to the utility
 - apparent power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

LiveView Harmonic Reactive Power Flows

The harmonic reactive power flows table displays real-time reactive power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic reactive power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.

You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic reactive power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- Remaining hold
- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
 - power flow in from the utility
 - power flow out to the utility
 - reactive power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

LiveView Harmonic Real Power Flows

The harmonic real power flows table displays real-time real power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic real power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.

You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic real power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- remaining hold

- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
 - power flow in from the utility
 - power flow out to the utility
 - real power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

LiveView Demand Current Summary

This summary displays a demand current summary for your system devices. Use the information from this table to help monitor the system's demand current.

After opening the demand current summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The demand current summary provides this data:

average demand current and peak demand (both in amps)

- Ia
- Ib
- Ic

Live View Demand Voltage Summary

This summary displays a demand voltage summary for your system devices. Use the information from this table to monitor the system's demand voltage.

After opening the demand voltage summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The demand voltage summary provides this data:

demand voltage

- Vab
- Vbc
- Vca
- Van
- Vbn
- Vcn

LiveView Viewer

Use this screen to view table templates, and to view or create table views, in the LiveView Viewer.

To open this screen, in Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 193](#)). In the PLS_Example project, there is a tab for LiveView. For information about an individual table, click a link from the Contents folder.

NOTE: If you plan to view a table using the [Rapid Access Labels on page 200](#) feature, do not change its name after you print the QR code. If the name is changed, you must generate a new rapid access label.

Open LiveView from a URL Link

Before you can open LiveView from a URL link, you must select a template and the desired devices, display the table, and save it as a View.

To open this view using a URL link, use one of these options:

- From the computer where LiveView is installed, enter
`http://localhost/LiveViewViewer`
- From a remote client computer, enter
`http://10.10.10.10/LiveViewViewer`
(where 10.10.10.10 is the URL of the server where LiveView resides)

To automatically open a specific table when you launch LiveView Viewer, add the table name to the end of the address. For example, to open the basic readings summary view while on the local computer, you would enter:

`http://localhost/LiveViewViewer/Basic Readings Summary View`

LiveView Viewer Display

The Live View Viewer displays with two tabs, Templates and Views.

Templates: A template includes all setup data (placeholders, formulas, thresholds, and formatting); but it does not have devices selected. The templates include those that are predefined (designated by the locked symbol: ), as well as those that have been defined in the Setup window.

To view a template, select it from the list. Then, select the device(s) for which you want to display values. (Only devices that have at least one assigned topic from the topic placeholders in this template are available for selection.)

Click Display Table to view the template in the right-hand pane.

To save a template as a view, display the template. Click () on the upper right of the Viewer page. In the View Name window, edit the name, then click OK. The new view is saved in Tables > Views on the server. The view will also display in the left-hand pane of the Views tab.

Views: A view is a template that is saved with its device selection(s). The views listed are saved on the server in Tables > Views. Views are available to all users, whether on the server or a client. They also display on the Views tab of the Live View Viewer.

To open a view, highlight its name, then click Display. The view displays in the right-hand pane with updated data. You can delete a view (click Delete, to the right of the View tab). You can change a view by adding or deleting devices and then either overwriting the view or saving it as a new view.

Update List: This link forces the cached table and view lists to be refreshed, displaying any newly added tables and views.

Select Device(s) and Update Device List: This link forces any new devices (with at least one assigned topic) to display. In the Select Device(s) list, you can move devices higher or lower in the list that you see, so that they display in the order you prefer. To do this, right-click and highlight a device, then click one of these icons:



: Move to the top (double arrow) or move up one step (single arrow)



: Move to the bottom (double arrow) or move down one step (single arrow)

[Where's My Device? on page 191](#): Click this link to explain why an expected device does not display in the table.

Template and View Features

The template (after you click Display) or view displays with devices and data. The following information is included:

Placeholders: All placeholders that were added during setup will display with the appropriate device name or tag value.

Thresholds: If any of the tag values are outside of the normal range established in the Thresholds that were added during setup, the font color will reflect the high or low status of that tag.

On the right-hand side of the top of the screen are these buttons:



Save (): Click to save a template as a "view." You are prompted to name the view (default: table name appended with "view"). The view is saved in Tables > Views on the server. The view includes the devices that were selected for the table.



Notes (): Click to display a description of the table that was added when the table was set up.



Print (): Click to print a copy of the table with its current values.

Last Update: This is the most recent date/time that the template or view values were updated.

Update Interval: Choose the interval of time that will pass between requests to update the data in the template or view. Options are:

Manual: Updates only occur when you click Update Now.

5 seconds

10 seconds

30 seconds

1 minute

10 minutes

Update Now: Click to manually update values and refresh the template or view.

Where's My Device?

This help topic displays when you click "Where's My Device?" below the device list in LiveView Viewer.

Missing Topics

Only devices that have topics available for the selected template will appear in the device selection list. If you do not see an expected device, it is missing because it does not include topics that are used in this template.

If this is a template that you have created, you can open the template in the [LiveView Placeholders on page 195](#) screen of LiveView Template Editor to add the placeholder(s). If this is a predefined template, you cannot change it; you will need to make a duplicate template and then add the desired placeholder(s).

Clear Cache and Platform Refresh

If the Schneider Electric CoreServiceHost has not been refreshed after devices or topics have been added, you should clear the cache and refresh the platform in order to access the new devices or topics.

See [Clear Cache and Refresh Platform on page 145](#) for instructions.

Set Up LiveView

Use this application to begin creating, duplicating, modifying, and deleting LiveView templates and views.

You can configure a LiveView template in LiveView Template Editor, and then display it on the server or on a web client. A "template" includes all of the setup data except devices.

You can create "views" of templates in LiveView Viewer. A "view" is a template that includes devices.

To access LiveView Template Editor, click Start > Programs > Schneider Electric > PowerSCADA Expert > v8.0 Config Tools > Template Editor.

Only one user at a time can access LiveView Template Editor. When a user accesses LiveView Template Editor, a file called *TemplateEditor.lock* is saved on the PowerSCADA Expert folder of the server (default location: Program Files > Schneider Electric > PowerSCADA Expert > v8.0 > Applications > LiveView > Viewer). If necessary, an administrator can unlock the utility by deleting *TemplateEditor.lock* from the server.

Here you can see:

- Notes icon (): This opens a free-form field to add any descriptive information about the template that will be useful. The information displays in a notes field, to the right of the template. This field closes when you click Done.
 - In the left-hand pane are the following:
 - New:** (You are prompted to save if you are editing a template that is not saved.) Click to save the template you are editing, and then to add a new template. The "New Template" name displays in the list, a new template file is uploaded to the server in Table > Templates, and an empty template displays in the right-hand pane. All fields are set to their defaults.
 - Duplicate:** Click to save a copy of the selected template. The current template name is used with "Copy" appended. Use this option to edit a predefined template.
- NOTE: If you find that a predefined table does not include enough cells for the data you want to display, create a duplicate. Then add the needed cells to the duplicate.
- Delete:** Click to delete the current template (you cannot delete predefined templates). Confirm that you want to delete it. All views associated with the template will also be deleted.
 - Select Template:** This list includes all of the templates that are set up. Predefined templates display a lock icon () to the left of the name. These templates cannot

be deleted or edited.

Template Name: Overwrite the current name, which updates the template here and in the list of templates. This will also update the views that are associated with this template.

Single Device/Multiple Device: Click the radio button for the type of template you want (default: Single Device).

View Area: Use this field to determine the area of the table that will be viewed in LiveView Viewer. When you set up a table, there may be information (such as formulas or notes) that you do not want to display in the final table in the Viewer. To select only the material that you want to view, do one of the following:

- In the View Area text field, type the cell range that you want to view (for example, A1:D20).
- Select the cells that you want to include, then press Use Selection.

In either case, a border displays around the cells in the range you select.

Save the template. When you view it in LiveView Viewer, it will only include the cells you selected.

Save: This button becomes live only after you make a change to a template that is being edited. The template is saved as an .xlsx file; it is uploaded to the server in Table > Templates. The saved template appears in the View tab after you click Save. (You do not need to click Save when you create a new template or a duplicate; these files are automatically saved.)

To create a new template, see [Create a LiveView Template on page 194](#).

To modify an existing template, see [Modify LiveView Template on page 198](#).

To duplicate a template, see [Duplicate LiveView Template on page 198](#).

To delete a template, see [LiveView Delete on page 199](#).

See also:

[LiveView Formulas on page 196](#)

[LiveView Placeholders on page 195](#)

[LiveView Thresholds on page 197](#)

[LiveView Formatting on page 194](#)

Create Menu Item for LiveView Page

The following steps describe how to interact with the LiveView application via the runtime environment.

1. From Citect Project Editor, click System > Menu Configuration.
2. Add the new menu item that you want for each of your LiveView tables.
3. In each of these menu items, in the Menu Command line, add the Cicode method that will display a LiveView tab. You can create your own custom method or use the default:

```
PLS_LiveViewDsp (STRING sIPAddress, STRING sViewName = "", STRING  
sTitle = "")
```

Example:

```
PLS_LiveViewDsp("10.10.10.10", "BasicReadingsSummary",  
"ShowConfiguration/MyConfiguration", "Basic Readings Summary");
```

which opens a configured LiveView table view with the saved configuration name "MyConfiguration".

Create a LiveView Template

To add a new LiveView template:

1. Open LiveView Setup, then click New.
An empty template displays with a "New Template" name.
2. Overwrite the template name with a name of your choice, up to 100 characters (limited to A–Z, a–z, 0–9, spaces, underscores, hyphens, and parentheses).
3. To continue setting up the template, click one of the links below.
Template Name: default: New Template (or, if an earlier New Template exists, appended with a 1)
Single Device/Multiple Devices. The default is single device. If you want a multiple device table, click Multiple Devices.

NOTE: Table grid lines do not display in the LiveView Viewer, but they do display in LiveView Setup.

See also:

To add data formulas to the real-time table, see [LiveView Formulas on page 196](#).

To add data (device names and tag names) to the real-time table, see [LiveView Placeholders on page 195](#).

To add visual alerts (color changes) when the value of the tag associated with a cell becomes too high or too low, see [LiveView Thresholds on page 197](#).

To add formatting to cells, such as font and font size, see [LiveView Formatting on page 194](#).

LiveView Formatting

Use this feature to add cell formatting, such as font, font size, and color. To use this feature, click the Formatting subtab in LiveView Template Editor. A formatting toolbar displays on the screen. It allows you to set the appearance of the cells in the template.

NOTE: Formatting changes become visible only after you click outside of the cell that you change.

To format a cell or range of cells, select the cell(s). When you select a format, the active cell(s) will be set to the specified format attribute. When a cell becomes active, the format selections on the toolbar will reflect the selections for that cell. When you select multiple cells, the format selections will reflect those of the first cell you select.

Then you can format the cell appearance:

- Choose font and font size.
- Choose whether to apply bold, italics, or underline.
- Choose a color for the font (default is black), and for the background of the cells (default is white).
- Choose the horizontal alignment: flush left, centered, or flush right.
- Choose the vertical alignment: top, center, or bottom.

- If more than one cell is selected, Merge Cells is enabled. Check this box to merge the selected cells into one large cell.
- In the Data Type drop-down box, select the type of data that will be in the selected cell(s):
 - **Text** (default); the *Wrap Text* box displays; check this box if you want text to wrap and stay within the cell.
 - **Date**; In the *Format* field that displays, type the format you want to use (Excel formatting is supported):
24-hour format: m/d/yy h:mm:ss
AM/PM format: m/d/yy h:mm:ss AM/PM or m/d/yy hh:mm:ss AM/PM
 - **Number**; In the *Decimal Places* field that displays, choose the number of decimal places you want; if desired, check the *Use 1000 Separator* box to insert the separator (for example, comma, depending on your regional settings).
- You can resize the row height or column width by dragging row/column header. A tooltip displays the height or width as you resize it.
Alternatively, right click anywhere in the template to display a context menu that allows you to insert/remove columns or rows, or to type the column width and row height.

Save your changes.

See also:

To add data formulas to the Live View template, see [LiveView Formulas on page 196](#).

To add data (device names and tag names) to the Live View template, see [LiveView Placeholders on page 195](#).

To add visual alerts (color changes) when the value of the tag associated with a cell becomes too high or too low, see [LiveView Thresholds on page 197](#)

See also:

[LiveView Formulas on page 196](#)

[LiveView Thresholds on page 197](#)

[LiveView Placeholders on page 195](#)

LiveView Placeholders

Placeholders provide the data (device names and tag names) to a LiveView template. The placeholders are the identifiers that are added when setting up the template, but are replaced with the name of the selected device or the tag value when the template is viewed.

To use this feature, click the Placeholder subtab in LiveView Template Editor. Then enter the appropriate information in the following fields.

1. To begin, place the cursor in a cell. Note that the Insert Location displays the cell number for the placeholder you are setting.
2. **TAG Value/Device Name:**
From the drop-down field in the top left corner of the page, choose either Tag Value or Device Name.
Tag Value: Select the tag group, such as Alarm, Current, Energy. Beneath the tag group, select the specific tag you want. The list is filtered to include only the most common tags that belong to the group you selected. To view all of the tags available in this tag group, check **Show Advanced**.

Device Name: The list of devices is filtered to include only devices for which this template's data is available. To display the device name in this cell of the template, select Device Name. You will choose the actual device during runtime.

3. **Insert Location:** This offers a second way of inserting the placeholder location. After choosing the device or tag, type the cell number for the placeholder cell.
4. **Insert:** Click to add the selected placeholder to the specified cell.

Continue adding placeholders as needed.

See also:

[LiveView Formulas on page 196](#)

[LiveView Thresholds on page 197](#)

[LiveView Formatting on page 194](#)

LiveView Formulas

Follow these instructions to add the formulas needed to include data in a Live View template. You can add formulas to:

- add, subtract, multiply, or divide the contents of two individual cells
- add, multiply, or average the contents of a range of cells

To use this feature, click the Formulas subtab in LiveView Template Editor. Then choose one of the following fields:

- **Cell:** Use this field to enable a formula for two individual cells. Then enter:
 - Cell 1 Address:** Type the cell address. The cell address displays in this field.
 - Operator:** Choose the operator you want to use: +, -, *, or /.
 - Cell 2 Address:** Type the cell address. The cell address displays in this field.
- **Cell Range:** Use this field to enable a formula for a range of cells. Then enter:
 - Operation:** Choose average, product, or sum.
 - Cell Range:** Type the cell range (format C4:C20), or select the range of cells to include in the formula. The cell range displays in this field.
- **Insert Location:** Type the cell number.
- **Insert:** Click this button to build the formula you have specified, and to add it to the cell you added to Insert Location.

Repeat the above procedure for the rest of the formulas you want to use for this Live View template.

NOTES:

- You must "Protect Current Sheet" in order for formulas to be maintained and visible in the LiveView Template Editor.
- If you want to use conditional formulas ("IF" formulas), you must first create them in Excel. To do this, you must access the template you want on the server (Program Files > Schneider Electric > Applications > LiveView > TemplateEditor > Templates Temp). Open the template in Excel and add the conditional formula(s) that you want. After you save the changes, the formula(s) will function correctly in Live View. You must copy the IF statement into every cell of the column that displays the result of the IF statement.
- In multiple device tables that rely on formulas to display information for each device, the results column will display zeroes when that row has no device in it. To

avoid this, use a formula that will display no result if there is no device in that row. In the following example, when no device is in cell A2, no results will display (no zeroes) in cell E2.

	A	B	C	D	E
1		Value 1	Value 2	Value 3	Sum
2	<<"Dn">>	<<"POWER:1039">>	<<"POWER:1040">>	<<"POWER:1041">>	=IF(ISBLANK(A2),"",SUM(B2:D2))

See also:

[LiveView Placeholders on page 195](#)

[LiveView Thresholds on page 197](#)

[LiveView Formatting on page 194](#)

LiveView Thresholds

Use this feature to display tag readings that fall outside of the normal range. You can apply it to an individual cell or a range of cells. You determine the tag or tags for which you want to display out-of-normal (threshold) readings. When the value of the tag in a cell (or any tag in a cell range) is below the minimum or above the maximum that you set, the tag value displays in the threshold cell.

You can set both minimum and maximum values for a cell or cell range. Use different colors to indicate the high and low readings.

To use this feature, click the Threshold subtab in LiveView Setup. To add a threshold:

1. **Cell:** For a single cell: Select the cell for which you want the font color to change. The font color will change when the value for the tag in that cell goes above the specified Max Value (or below the Min Value) for the threshold.
or
Cell Range: For a range of cells, either select the range, or type the range in the format C4:C20.
- NOTE: When setting up a multiple-device table, you should use a cell range to ensure that threshold font colors display for each device in the table.
2. **Min Value:** Type the low value for the "normal" range. If the tag value drops below this value, the cell font color will change as specified in step 3.
3. **Below Min Threshold Color:** Open the color palette and select the font color that you want to indicate the "low" status.
4. **Max Value:** Type the high value for the "normal" range. If the tag value goes above this value, the cell font color will change as specified in step 5.
5. **Above Max Threshold Color:** Open the color palette and select the font color that you want to indicate the "high" status.
6. **Insert Location:** Choose an empty cell, one that is not part of the table. This cell will be the location for the threshold definition that you are creating.

The default cell for the threshold definition is the next available cell in the template. For example, if the tag in cell B7 has an unused cell to the right of it (C7), the threshold definition defaults to C7. Then, when the value in B7 exceeds the threshold defined in C7, the value in B7 displays in the font color you specified. To override the default cell location, change it in the Insert Location field.

7. **Insert:** Click here to create the threshold(s).

The threshold definition is in the form: <<Threshold;B2:B20;Min=100;Max=1000>>

See also:

[LiveView Formulas on page 196](#)

[LiveView Placeholders on page 195](#)

[LiveView Formatting on page 194](#)

Modify LiveView Template

You can modify any template except one that is predefined. Predefined templates have a lock icon () beside their names.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 193](#)). In the PLS_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to modify.
The template displays.
3. You can change any field on the template. Click any of the subtabs (Placeholder, Formula, Threshold, or Formatting) to edit the related information. For help on the subtabs, see the "See Also" links below.
4. When you have finished making changes, click Save.

Continue working with other templates.

See also:

[LiveView Placeholders on page 195](#)

[LiveView Formulas on page 196](#)

[LiveView Thresholds on page 197](#)

[LiveView Formatting on page 194](#)

Duplicate LiveView Template

Use this feature to make an exact copy of any template, including predefined templates. The duplicate will not be locked, allowing you to edit and save it as a different template.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 193](#)). In the PLS_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to duplicate.
The template displays.
3. Click Duplicate (on the top of the left-hand pane).
The duplicate template is added to the list. It has the same name of its original template, appended with "Copy."
4. Change the name of the duplicated template to differentiate it from its original.
5. Make the desired changes and then click Save to save them.

LiveView Delete

You can delete any template except one that is predefined.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 193](#)). In the PLS_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to delete.
The template displays.
3. Click Delete (on the top of the left-hand pane).
You are prompted to verify the deletion.
4. Click Yes to delete the template, or click No to cancel the deletion.

Continue working with other templates.

LiveView Template Localization

You can easily localize your LiveView templates. To do this, you will need to replace English terms with the localized ones. Follow these instructions to create the localized file:

1. Choose the language that you need for your localized terms.
 - Go to Control Panel > Internet Options > Languages. In the Language Preference box, click Add to display the possible languages.
 - Highlight the language you need. For example, French for France is: **French (France) [fr-FR]**. Click OK.
 - In the Language Preference box, highlight the new language and click Move up to move it to the top of the list of languages. Click OK.
2. Locate the file called LiveViewViewer.resx. The default location is in the App_GlobalResources folder.
3. Make a copy of the resx file.
4. Use Visual Studio to open the copy resx file.
5. Replace the English terms in the right hand column with the localized terms that you need.
6. Save this file with a new extension, which must match the language. For the example above, the new file name would be:

LiveViewViewer.fr-FR.resx

The LiveView template fields that you replaced will now display in the localized language.

Enable Windows Authentication for LiveView

You may wish to use Windows Authentication for logging in to LiveView. If you want to use Windows Authentication, you must follow standard IIS authentication methods. Follow the steps below to configure Windows Authentication.

NOTE: These steps are specific to Windows 7; they may be different for other operating systems. For further assistance, view Microsoft's documentation on this topic at:

[http://technet.microsoft.com/en-us/library/cc754628\(v=ws.10\).aspx](http://technet.microsoft.com/en-us/library/cc754628(v=ws.10).aspx)

1. Turn on the Windows Authentication feature:

- a. From the control panel, click Programs and Features > Turn Windows features on or off.
 - b. Check Windows Authentication.
2. Enable Windows Authentication in IIS:
 - a. From the Control Panel > Administrative Tools, choose Internet Information Services (IIS).
 - b. Select the root node from the tree on the left (or the LiveViewViewer node, if this server hosts multiple sites).
 - c. From the right hand pane, in the IIS section, click Authentication.
 - d. Enable Windows Authentication.
3. Modify web.config to specify Windows Authentication:
 - a. Locate web.config:
C:\Program Files\Schneider Electric\PowerSCADA Expert\v8.0\Applications\LiveView\Viewer
 - b. Change the line:
<authentication mode="Forms">
to
<authentication mode="Windows">
4. Add roles to web.config:

You must add roles to web.config to allow access to the LiveView application. For example, to allow the role (group) Administrators, add the following to the web.config file:

```
<authorization>  
  <allow roles="Administrators"/>  
  <deny users="?" />  
</authorization>
```

Modifying the web.config file is an advanced topic that is covered on the Microsoft website.

You will enable Windows Authentication in IIS and modify web.config to specify Live View.

See the following link for instructions provided on the Microsoft website:

<http://www.iis.net/configreference/system.webserver/security/authentication/windowsauthentication>

Additional information may be available in the following knowledge base article:

<http://support.microsoft.com>, and search on kb/815179.

Rapid Access Labels

Use this report to create quick-response code (QR code) stickers that can be placed on your system equipment to provide quick access to PowerSCADA Expert standard reports and LiveView table views. You can also generate a label for any URL. After you create and print the code stickers, you will be able to read them with a QR code reader that is installed on a Smart Phone, iPhone, or other device.

Before You Begin

Make sure that you have completely configured your system. This includes:

- Set up all servers, equipment, and addressing

- Create the reports and LiveView views that you want to see. Note that the report configuration names and Live View view names cannot be changed after you create the QR stickers, as the links would be broken to the reports/table views.
- To view a report, you must first save the report configuration. See [Run a Report and Save its Configuration in Create and View Basic Reports on page 171](#) for instructions.
- All devices that will be used to scan QR codes must be on the same network with the server.
- Purchase the label stock paper for the labels you will print. Use **Avery 6578 Label Stock**, or equivalent. Other label stock may not be spaced correctly, which could result in the labels not printing correctly.

Create the Sticker

1. Ensure that you have a laser printer set up and available for printing.
2. At the printer that you will use to print the labels, insert the blank label stickers. Use **Avery 6578 Label Stock** or equivalent.
3. Browse to the Rapid Access Labels report in the reporting web application. When prompted, enter your Citect user account information. Click Login.
The Rapid Access Labels screen displays.
4. From the *Server Address* drop down list, choose the IP address that is connected to the same network as the wireless access points.
Do not use an IP provided by DHCP, as the IP address can change frequently.
If your network supports DNS, we recommend that you use the machine name of the server.
5. In the *Port* box, accept the default "80" or, if necessary, enter a different port.
6. In the *Select items to generate labels* box, check the report configuration(s) and LiveView table(s) for which you want to print stickers.
7. Optional: You may wish to print a sticker for a different URL (such as a corporate website). To do this, enter the URL in the *URL* line of the *Manual URL Entries* box (the site name automatically displays in the upper box).
8. Optional: On the *Caption* line, you can type any text that you want to have printed above the QR code on the sticker. If you want the output table or report to have a title, enter it here.
9. Click *Generate Report*.

NOTE: To print correctly, use the icon on the report control bar, not the one from the browser (which would add a header and footer, and throw off alignment).

Read the Sticker

The sticker(s) print at the designated printer. Each one has a title that is one of the following:

- a report configuration name
- a LiveView name
- user-entered text from the *Caption* text box

Place each sticker in the desired location, such as next to the device that is being monitored.

To read a sticker, use a QR code reader (such as a Smart phone or iPhone). The reader must have access to the network and server. We recommend that you use the QR Droid application if you are viewing reports/tables from an Android phone.

Troubleshooting

If you cannot read the QR code, verify the following:

- Your phone/reader has access to the wireless network, and the server can be reached by the IP you selected when generating labels.
- The server address and port name are correct.
- The report configuration name or LiveView table name are correct (and have not been changed or deleted).

Multi-Monitor Support

PowerSCADA Expert 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, 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

The windows display 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

The Demo Mode

You can run PowerSCADA Expert without the hardware key in demonstration (Demo) mode. In demonstration mode, you can use all product features normally, but with restricted runtime and I/O.

If you configure PowerSCADA Expert to run as multiple processes on one CPU or multiple CPUs, you cannot use it in demo mode; but if you run it as one process, you can use a demo mode. The following demonstration modes are available:

- The one-line engine will run for 60 minutes.
- Reporting and LiveView do NOT work, because it requires a valid license.
- 15 minutes with a maximum of 50,000 real I/O.
- 10 hours with a maximum of one dynamic real I/O. This is useful for demonstrations using memory and disk I/O. PowerSCADA Expert starts in this mode if no hardware key is available. If the system detects that you are using more than one real I/O point at runtime, it will switch to the 15-minutes demo mode.

Writing to any tag through DDE, CTAPl, or ODBC will cause that tag to contribute to the dynamic point count, even if it is a memory or disk I/O point. So, if you write to more than one point through these interfaces, it will switch to the 15-minute demo mode.

- 8 hours with a maximum of 42,000 real I/O. This is only available through special PowerSCADA Expert Integration Partners (CIP) keys.

Application Configuration Utility

Use the Application Configuration Utility to easily configure many features that would require more time-consuming effort if performed via INI settings. See [Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#) for details about these settings.

Options that are available on every page are:

Restart Services: This feature is available only when changes are made to the Citect Data Platform page. Click this link to restart Application Services. If you click inside any field of this area and then click Display Selected or Display All, you are prompted to save your changes before the configuration file displays.

Display Selected: Click this link to display the settings that have been entered in specific area of the Application Configuration Utility (Application Services, Application Services Host, Applications, Security) that you are viewing.

Display All: Click this link to view the settings that have been entered for the entire Application Configuration Utility.

Tooltips: To view help for an individual field, point your mouse and hover over the field.

The Application Configuration Utility includes these sections:

Application Services: The [Application Services on page 204](#) section includes screens for setting up:

- [Diagnostics on page 204](#)
- [EcoStruxure Web Services Setup on page 204](#) (EWS)
- [Event Notification Service on page 205](#)

Application Services Host: This includes:

[Application Services Host—Citect Data Platform on page 205](#): four tabs contain settings to configure server/user name, ignored devices and topics, deadbands, and to display the license of the server.

Applications: This includes:

[Basic Reports on page 207](#): Use this screen to set up the delivery mode and email address from which PowerSCADA Expert 8.0 basic reports will be sent.

[One-Line Engine on page 206](#): Three tabs contain settings to configure one line engine behaviors.

Security

- [Single Sign On on page 208](#)

Application Services

Use this section to configure services that connect with Citect. There are three tabs:

[Diagnostics on page 204](#): This is a read-only list of web services hosted by the Schneider Electric CoreServiceHost, details about them, and whether they are running. You can also choose the logging level.

[EcoStruxure Web Services Setup on page 204](#): EWS uses web-based http protocol to transfer data. It enables two-way data transfers, which allows the acknowledgment of alarms from SBO.

[Event Notification Service on page 205](#): This feature serves alarms and events to the Event Notification Module application through a web service endpoint named Alarms/Alarm Proxy.

Diagnostics

Logging Level:

This feature turns on extra diagnostic information that can be useful when diagnosing problems that occur in application services or its hosted applications (such as LiveView). Choose the level of logging to be used in all applications. Debug and Verbose increase the amount of information that is logged during runtime for applications such as Basic Reports and LiveView.

- Normal: Use when the project is live.
- Debug: includes additional logging statements (in the Windows event log named PowerLogic). This logging should not affect performance in the system during runtime.
- Verbose: releases additional diagnostic information, such as large lists, that could affect system performance.

Service Inventory:

This is a read-only list of web services hosted by the Schneider Electric CoreServiceHost, details about them, and whether they are running.

EcoStruxure Web Services Setup

This feature configures the PowerSCADA Expert EcoStruxure Web Services (EWS) server. (See [EcoStruxure WebServices \(EWS\) on page 169](#) on page 1 for a description of this server.). Do not confuse this information with the EWS server that was released with PowerSCADA Expert 7.40. That implementation is specific to the Citect core. It was developed only for real-time tag data acquisition. The implementation being released with this product also acquires historical data and alarms.

The fields are:

- Alarm Acknowledgment Wait Period: The amount of time allowed for PowerSCADA Expert to process an alarm acknowledgment request. Choose a value that allows the system enough time to allow acknowledgments to be processed, while not so long as to delay processing.
- Initial Alarm Request Length: The number of days worth of alarm to request from PowerSCADA Expert.

- **Max Request Size:** The number of alarms returned with one request. The default (1000 alarms) should be sufficient to maintain alarm data integrity (ensuring that all alarms are returned in each call), while also maintaining system performance.
- **Alarm Settle Time:** The number of seconds "grace period" to allow the Citect Alarm Server to finish inserting alarms that are in process at the time of the poll. If you set this too low, you could miss alarms. If you set it too high, it may take longer for alarms to come into EWS.
- **EWS/Citect User Association:** Use this block to manage user names and passwords. This provides EWS Digest Authentication for the user, permitting them to view data. However, for the user to be able to acknowledge alarms, the username/password must match a username/password added to the PowerSCADA Expertv8.0 project. When this user acknowledges an alarm through EWS, Citect verifies the credentials of the user and acknowledges the alarm under this user's identity.

To add a user, click Add. At the Add User screen, type an established Citect username and password. Click Test Citect Credentials to verify the name and password. When you enter a valid username and password, a message displays telling you they are valid.

Avoiding EWS Provider Timeouts

When you attempt to retrieve a large amount of data from the EWS server, you could see an error telling you that the call has timed out. To correct this, you can temporarily increase the timeout period, which will allow the target application to receive the data. To do this, modify the key named *ProviderTimeoutInMinutes*, found in the EWS virtual directory, under Web.config.

Default location;

C:\Program Files (x86)\Schneider Electric\PowerSCADA
Expertv8.0\Applications\EWS\Web.config

After the data that you want is processed, you need to return this key to its original setting.

Event Notification Service

This feature serves alarms and events to the Event Notification Module (ENM) application through a web service endpoint named Alarms/Alarm Proxy. The fields are:

- **Maximum Number of Alarms/Events per Call:** This field controls the maximum number of alarms or events that can be requested in one poll. As the requesting client continues to request data, additional data will be served at the same rate.
- **Poll Period:** The frequency at which the Event Notification Service polls Citect runtime for alarms (in seconds).
- **Alarm Settle Time:** The number of seconds "grace period" to allow the Citect Alarm Server to finish inserting alarms that are in process at the time of the poll. If you set this too low, you could miss alarms. If you set it too high, it may take longer for alarms to come into ENM.

Application Services Host—Citect Data Platform

This section relates to how the core service host connects to Citect.

Use this page to link a Citect user name and password to be used when the Schneider Electric CoreServiceHost services connect with Citect runtime. Before you begin:

- Add the username/password to the Citect project.
- Have Citect running in runtime mode.

Then follow these steps:

1. **Citect I/O Server Address:** Choose the server address for the project that is running.
2. **Citect User Name:** Type the user name for this user.
3. **Citect Password:** Type the password for this user.
4. **Test Credentials:** Click to verify these credentials. If you see an error, verify the name and password, and that Citect runtime is running, and try again.

When your Citect project is running and the credentials are valid, you see Connection Successful. The user name and password can be used to connect to Citect.

Citect Licensing Details: This is a read-only field that displays the license key currently in use on the Citect server machine.

Applications

Use this section to set up information for:

[Basic Reports on page 207](#)

[One-Line Engine on page 206](#)

One-Line Engine

To access this feature, click Start > All Programs > Schneider Electric > PowerSCADA Expert 8.0 > Config Tools > Configurator. There are three tabs in the One-Line Engine module. On all three tabs, two buttons at the bottom allow you to:

Restart AOL: Click to manually restart the Advanced One Line engine.

Save: Click to save the settings you have entered.

Citect User: After you add a user to your Citect project, use this tab to test whether the user ID can be used by the One Line engine to connect with runtime. Type the Citect user ID and password, and click Test Credentials. The test connection will attempt to log in with this user information. A message displays, telling you whether the user information passed. If it does not pass, you see a message telling you that the connection failed because the user name/password are incorrect or PowerSCADA Expert is not running. Make sure that PowerSCADA Expert is running and that the user name/password have been set up in Citect, then try again.

General: You can edit the following parameters that enable one lines to run properly. For more complete descriptions of the parameters, see [Add INI Settings to AdvOneLine.ini and Citect.ini on page 121](#). If you are not setting up a redundant system, the default settings should be sufficient.

- Update Interval: interval in seconds at which the system tries to solve the system one-line
- Max Startup Delay: sets the amount of time in seconds the AdvOneLine.exe has to start up
- Health Timeout: performance parameter; dictates the amount of time in seconds that must elapse before the one-line engine is considered to be non-functioning
- Log File Length: suggests the log file length in number of lines
- Debug Level: selects the level of logging for AdvOneLine.exe

Redundancy:

- Primary Server IP: used in redundant configurations to specify the IP address of the primary IO Server. Click Clear to clear the current address, then type the correct address for the primary server.
- Standby Server IP: used in redundant configurations to specify the IP address of the standby IO Server. Click Clear to clear the current address, then type the correct address for the secondary server.

Use the following two buttons to save an encryption key and export it to another computer as an .aes file.

NOTICE

LOSS OF SYSTEM CONFIDENTIALITY/INTEGRITY/AVAILABILITY

Do not save the .aes encryption file to the local computer. Always export it to a secure site, such as a secure network or to the target computer.

Failure to follow these instructions can result in unauthorized access to, modification of, and unavailability of the system.

This allows you to move an INI file from one computer to another and to have its contents unencrypted for use by that computer.

- Export Key: After making/verifying changes here or in the AdvOneLine.ini file, click to save a configuration that you can use on another computer.

A Save As window displays, allowing you to browse to the preferred location. Save the .aes to a secure location, such as a secure network drive or a flash drive.

- Import Key: After you save the .aes file to the secure drive, ensure that the drive is accessible to new computer. At the new computer, click this button to access the .aes file.

After you access the .aes file at the new computer, copy the INI file to the new computer. You will be able to access and use it.

Remove the .aes files from the source computer.

Basic Reports

Use this screen to set up the delivery method and email address from which PowerSCADA Expert 8.0 basic reports will be sent. These settings specify the SMTP server for emailing basic reports. They are not used for Event Notification Service.

Define the following:

- Timeout: The number of seconds PowerSCADA Expert will attempt to deliver an email before no longer attempting
- Delivery Method: Network (default), Pickup Delivery from IIS, or Specified Pickup Directory. This is an SMTP-specific setting. In most cases, use Network. For more information on SMTP, see the Micrologic Developer Network website.
- 'From' Address: the address from which reports will be sent.
- Host: The IP or network address of the SMTP server.
- Port: The network port to be used; default for SMTP is 25.
- Use Default Credentials: If required by the SMTP server being used, uncheck the box and enter the appropriate user name and password. If not required, check the box and enter the SMTP user name and password used for reporting.

Single Sign On

Use this feature to associate a Citect user with either a PowerSCADA Expert (PSE) or Power Monitoring Expert (PME) username/password. When the Citect user is logged in to runtime and accesses an external applications (such as Dashboards), the SSO user password from PME is used to authenticate with the external application.

We recommend, when you use Single Sign On, that you maintain the components on the same computer or in a secure network. If higher security is needed, use SSL (https).

NOTICE

LOSS OF SYSTEM CONFIDENTIALITY/INTEGRITY/AVAILABILITY

Ensure that all single sign on information is secured, either on the same computer or a secure network.

Failure to follow these instructions can result in unauthorized access to, modification of, and unavailability of the system.

The fields are:

- **Application:** From the drop-down list, choose the application (such as Dashboards, Basic Reporting, Advanced Reporting, Diagrams, LiveView) for which you want Citect-to-SSO user mapping.
- **Timeout:** The time after which the system will stop trying to find a match. If no match is found, single sign on for this user will not take place.
- **Guest User:** This feature allows you to create users in PME or PSE that will have fewer feature privileges. The guest user does not have an established Citect account, but will need to have a PME or PSE account that matches this username/password.

Check Guest User, then click Edit to launch the Edit User dialog. In the Edit User dialog, type the SSO user and password that match the username and password of the PME or PSE user.

- **Users:** Add/modify the users to have access to the selected application. Use this area to add users who do need to have a Citect user account.
 - **Citect User:** the Citect username that is requesting authorization.
 - **SSO User:/SSO Password:** The established credentials for this user, either from PowerSCADA Expert or Power Monitoring Expert.

Single Sign-On (SSO) Calls from a Web Client

PowerSCADA Expert automatically detects calls that are made from a web client. The calls are sent to an I/O server. However, there are two additional requirements:

- You must first set two INI parameters for the I/O server and cluster to handle the call:

[SSO]

```
RemoteCallHandlerServer = {Name of I/O server that will  
execute call}
```

```
RemoteCallHandlerCluster = {Cluster of I/O server that will  
execute call}
```

- When using single sign-on from a web client, **the user must be logged in with an SSO account.**

Assign and Control 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:

[Default User Access Settings \(Privileges\) on page 209](#)

[Add and Modify User Accounts on page 210](#)

[Change Access Rights on page 211](#)

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 PLSecurity.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 PLSecurity.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_CBCControl	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 [Change Access Rights on page 211](#) 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 PowerSCADA Expert 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.

Add and Modify 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 user access, follow these steps:

1. From the PowerSCADA Expert 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 209](#) 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.

Use Windows Integrated Users

Instead of using the above method to define users in the project, you can incorporate PowerSCADA Expert users and security options with the standard Windows security system. Using the integrated Windows security feature, the Windows user can log on to PowerSCADA Expert 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 PowerSCADA Expert "role," which is defined in the project with associated privileges.

To assign Windows integrated user access,

To link a Windows user to a PowerSCADA Expert 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 PowerSCADA Expert 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 PowerSCADA Expert using any Windows user credential that is a member of the linked group.

When the name of a PowerSCADA Expert user also has the same name as a Windows user, the PowerSCADA Expert user takes priority at runtime. However, if a valid PowerSCADA Expert user login does not succeed 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.

For more information, see **Using PowerSCADA Expert with Windows Security** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

Change 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 PLSecurity.ci file (in the PLS_Include project).

To change access rights:

1. Open PLSecurity.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 PLSecurity.ci.

NOTE: PLSecurity.ci is overwritten any time that the PLS_Include project is overwritten. This happens with every release of the product, 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.

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 PowerSCADA Expert. These genies use a particular naming convention, which is described in the table below. In the PowerSCADA Expert 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 indicates PowerSCADA Expert 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 PowerSCADA Expert 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

Genie Abbreviation	Description
sl_closed_lv	closed low-voltage circuit breaker
sl_conductive_path_1	conductive path 1
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

Additional definitions:

bus	=	busway
cb	=	circuit breaker
hv	=	high voltage
lv	=	low voltage
dr	=	drawout
nd	=	non-drawout
fd	=	earth at bottom (feeder)
inc	=	earth at top (incomer)
nes	=	no earth
nc	=	not remote control
c	=	remote control

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

Genie Abbreviation	Description
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
lv_cb_nes_nd_nc	low voltage non-drawout circuit breaker, no earth, no remote control

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
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

Genie Abbreviation	Description
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 genres 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
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
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

Genie Abbreviation	Description
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	PowerSCADA Expert circuit monitor
egx	PowerSCADA Expert EGX
generic_meter	generic meter
ion_7650	ION 7650 meter
micrologic	all Micrologic meters
power_meter	PowerSCADA Expert power meter

Genie Abbreviation	Description
quantum	PowerSCADA Expert 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

Appendix 2: 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.

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:

[General PowerSCADA Expert Parameters on page 222](#)

[Performance Tuning Parameters on page 229](#)

[Waveform Parameters on page 237](#)

[Sepam Event Reading Parameters on page 238](#)

[MicroLogic Modules Configuration Parameters on page 239](#)

[Data Replication Parameters on page 240](#)

General PowerSCADA Expert Parameters

The following parameters are common to all PowerSCADA Expert devices.

watchtime

Controls how often the product 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
```

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 that the device is offline or communication has been lost with 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
```

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 PowerSCADA Expert 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]

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

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

TimeSync

Enables/disables time synchronization for the PM5000S driver. On startup and on a 15-minute schedule, the driver reads each device clock. If a device clock is not within the specified 10-second drift, the driver sets the time on that device to the current system time.

Parameter type: boolean

Default value: 0 (PM5000S) or 1 (PM5000S1)

NOTE: This is a driver-level parameter, not a protocol-level parameter. All entries must be under the PM5000S section of the .ini file. By default, the PM5000S1 protocol enables time sync. For the PM5000S, it is disabled by default because most devices will have battery backup and GPS time sync availability.

Example:

[PM5000S] TimeSync = 1

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 (2 for Sepam) (PM1200 requires that this value be set to 3911)

Examples:

[PWRMODBUS]

statusRegister = 1000

[PWRMODBUS.MYCLUSTER.PORT_DEVICE_PM1200]

statusRegister = 3911

StatusRegistersCount

Defines the number of registers 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: number of registers

Default value: 1 (PM1200 requires that this value be set to 2)

Examples:

```
[PWRMODBUS]
```

```
statusRegistersCount = 2
```

```
[PWRMODBUS.MYCLUSTER.PORT_DEVICE_PM1200]
```

```
statusRegistersCount = 2
```

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.

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 be 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

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.

Alarm Parameters

The following parameters are used for alarms.

UsePLSFilter

Controls whether alarm/event filtering is done by the PLSCADA filter form or the Citect filter form. Both forms cause the same information to display on the page, but each is presented in a different format.

Parameter type: integer

Default value: 1 (PLSCADA filter form)

Example: [ALARM] UsePLSFilter = 1

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

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

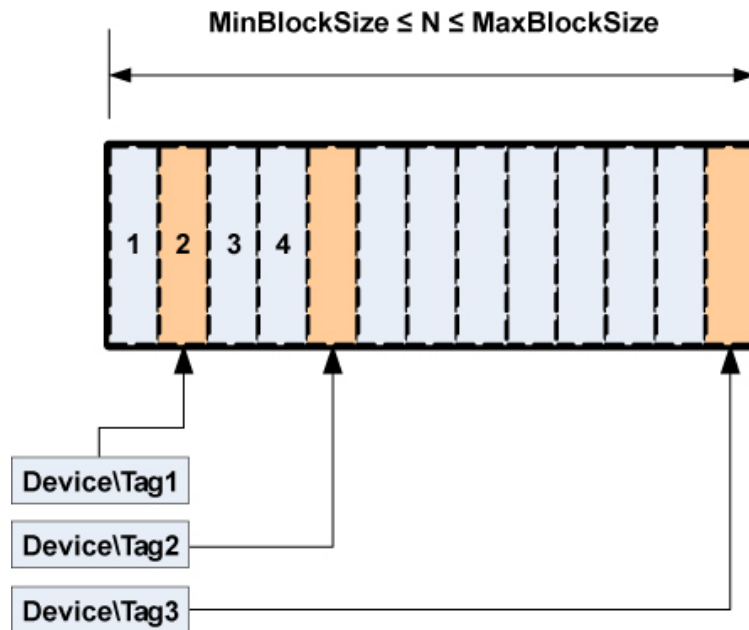
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_B]

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 how 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 $\geq$ PercentBlockFill, the block builder creates a multi-register block.



Parameter type: percentage

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

Normal Priority Tag Refresh: 1000 ms

High Priority Tag Refresh: 500 ms

Example:

[PM870.MYCLUSTER.PORT_1]

HighScanRate 25

LowScanRate 500

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.

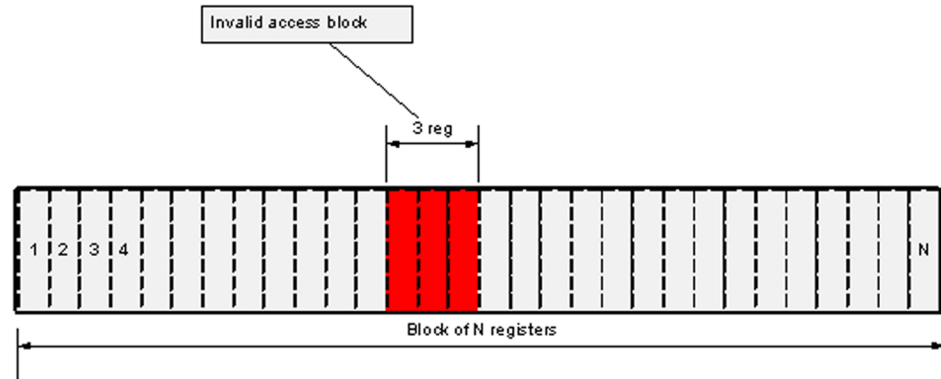
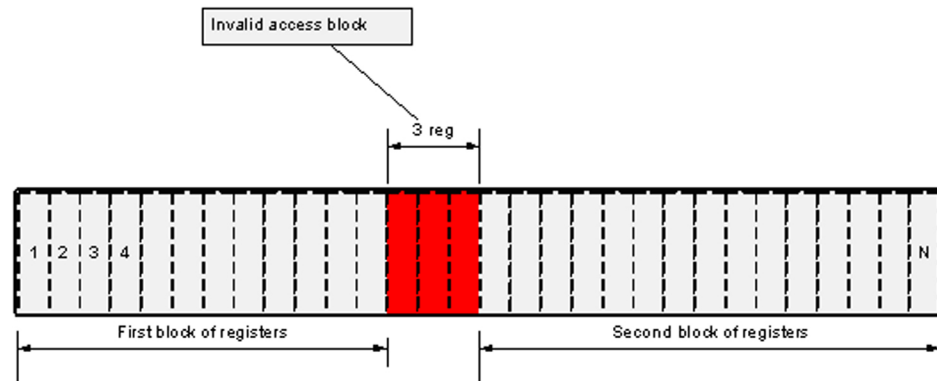


Figure 1 shows how the "invalid memory access block" affects MODBUS register blocking. In this situation, if the driver does not know that the block of 3 inaccessible registers exists, it will try to block all registers from 1 to N (depending on data that was requested by the real-time data collector). 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:



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

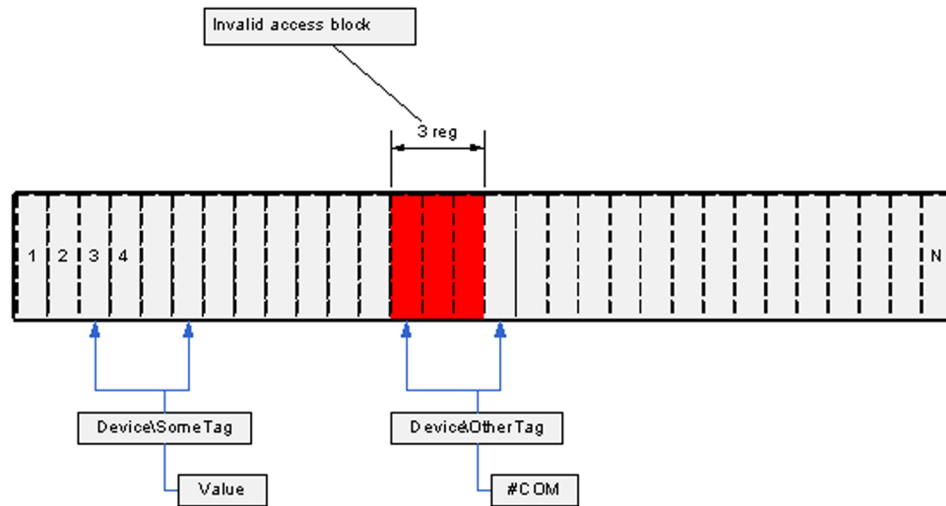
Example:

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:



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

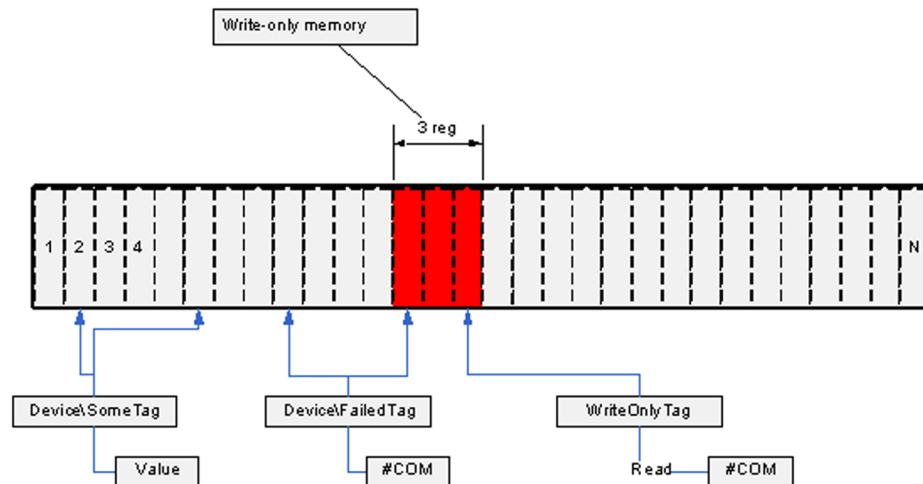
[ERROR] [MISC] [RealTimeData::BlockBuilder::AddDataPoint()] Cannot
add datapoint, one or more invalid memory addresses fall into non-
splittable block

[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.

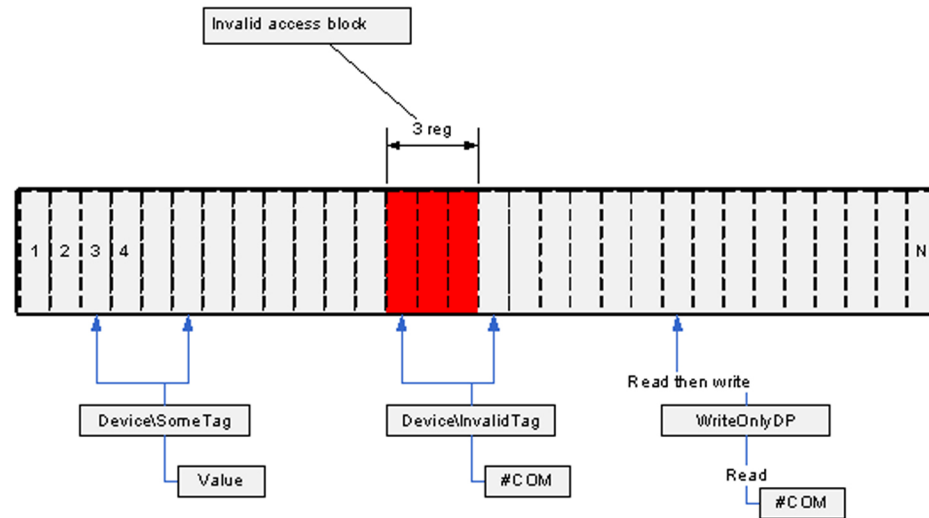
Write-only Memory

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:



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 not succeed.

Figure 5 illustrates a write-only datapoint:



Tag Blocking Notes

The drivers support an advanced blocking mechanism for tags. That is, real-time tags are no longer blocked together with write-only tags.

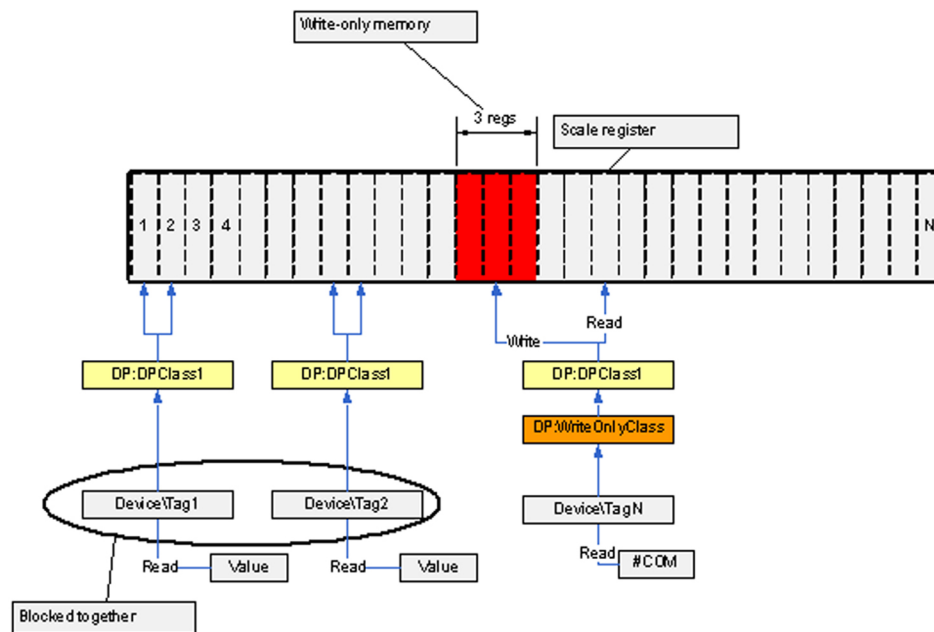
Tags found invalid, due to intersection with invalid memory areas, are not blocked with “good” real-time tags and will not therefore interfere with them.

Write-only Tags

Beginning with driver version 2.0.1.1, the write-only tags feature is fully supported.

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.

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 (such as clusters, individual devices)

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 the product 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)

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

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

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.

MicrologicV INI Settings

The MicrologicV device driver includes these additional INI settings:

- Level3: This is the level 3 device password (four digits), used by the driver when executing commands.
- Level4: This is the level 4 device password (four digits), used by the driver when executing commands.

If you do not supply this parameter, the driver uses the default device passwords.

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, the PowerSCADA Expert[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.

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\PowerSCADA Expert 8.0\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 PowerSCADA Expert 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]

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 alarms to indicate that communication was not successful 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

Integration Parameters

The following parameters deal with single sign-on and integration of PowerSCADA Expert and Power Monitoring Expert.

[SSO]PSEHostName

If you do not have Power Monitoring Expert installed, and you want PowerSCADA Expert reports, use this parameter. This parameter specifies the IP address for PowerSCADA Expert.

Default: localhost

[SSO]HostName

This parameter specifies the IP address for Power Monitoring Expert.

Default: localhost

[SSO]RemoteCallHandlerServer

This parameter specifies the I/O server that will execute the call from a web client.

Default:N/A

[SSO]RemoteCallHandlerCluster

This parameter specifies the cluster of the I/O server that will execute the call from a web client.

Default:N/A

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

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 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 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 project. Although not difficult to use, the person working in Cicode must have received Cicode training.

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.

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. PowerSCADA Expert 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 PowerSCADA Expert to achieve 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 the product without a hardware key. You can use all of the product features, but with limited runtime and I/O options. See [The Demo Mode on page 202](#) 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 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 PowerSCADA Expert to interact with a device or series of devices. PowerSCADA Expert 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 67](#) 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.

ICD file

IED capability description: This is the file that is imported into 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 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 PowerSCADA Expert, 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 PowerSCADA Expert, 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 PowerSCADA Expert. These alarms and associated waveforms can be read and displayed in the Alarm Log.

parity

Parity is used as a simple means of detecting error by verifying that the result is odd or even. In PowerSCADA Expert, parity is required for the generic serial or MODBUS/RTU comms methods, when adding a device.

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 PowerSCADA Expert.

point (see SCADA tag)**polling priority**

When adding a custom tag, this field determines the level of priority that PowerSCADA Expert uses when reading data from the related device. Options are low, normal, or high.

PowerSCADA Expert 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 PowerSCADA Expert, for use in creating graphics pages.

Profile Wizard

This tool allows you add device profiles to, or delete them from, a project. From the Profile Editor, you export profile data into a file that can be used in the project. From there, you use the Profile Wizard to add the device profile into a 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.
- The project **myCM4Project** instantiates the **myCM4Profile** and calls it **myModbusCM4**, and it specifies that it uses the Modbus driver.
- When this project is imported into the SCADA system, Modbus addressing will be used.

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 Expert 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.

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.

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 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, If 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 209](#) 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.

Appendix 5: Frequently Asked Questions (FAQs)

The following items provide information about topics that generate frequent questions.

If I don't use PowerLogic drivers, how do I create device profiles?

The easiest way to do this is to:

Create a device type using a non-PowerLogic driver (like MODNET).

1. Using that device type, create a device profile.
2. 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.

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.

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.

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 [Import Files into the Profile Editor on page 57](#) 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 PowerSCADA Expert.

What are the requirements for device names?

Device Name:

Keep Device name $\leq$ 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.

- 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.

How do I troubleshoot device communications issues?

PowerSCADA Expert drivers provide default communication settings that work with most devices. However, in cases when communication losses occur, 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 PowerSCADA Expert 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 PowerSCADA Expert 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. PowerSCADA Expert 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 PowerSCADA Expert, 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 TCP/IP: 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 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 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

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 graphics objects (such as circuit breakers and alarm pages) are constructed using a pre-defined set of user privileges the 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 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 209](#). For additional information, see **Using Security** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.0 > bin).

How do I manage projects in the Citect Explorer environment of Power-SCADA Expert?

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:

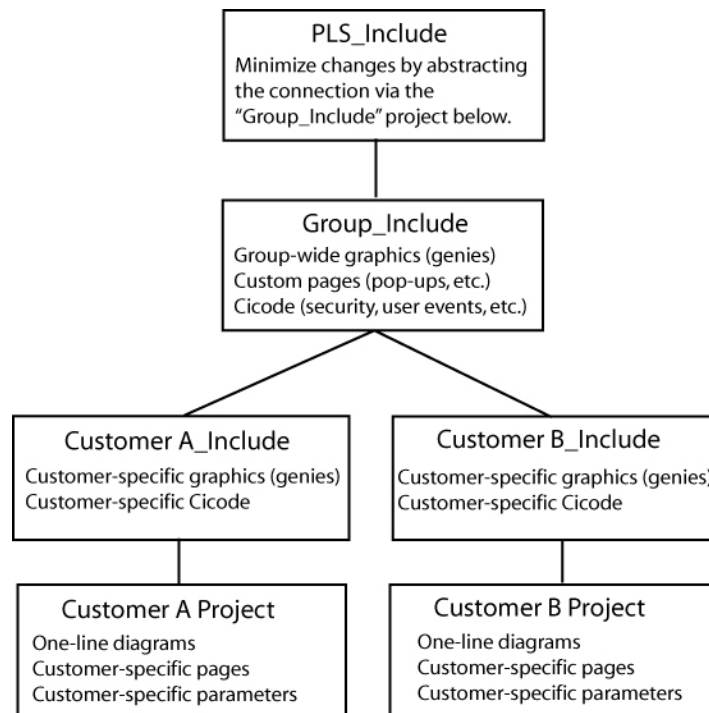
1. Keep original 'Master' copies of the PLS_Example and the PLS_Include projects for reference.
2. The Services group may develop a group-wide "include" project that will act as a conduit between the PLS_Include project and all customer projects (e.g., "Group_Include"). This will make the upgrading of 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.

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.

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 system, and that it preserves this feature:

- 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.

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 241](#))

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 ([View Device Profiles on page 39](#), "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 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 [Set a New Page as the Project Startup Page on page 112](#) 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).

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. 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\zav > 120).

If the expression is TRUE, the ACTIVE color will be shown. The active color is determined by the voltage level assigned.

If the expression is FALSE, the DE-ENERGIZED color 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 = All

DebugLevel = All (or Error)

LogFileArchive = Deprecated; no longer used. Use *[Debug]SysLogArchive* instead.

LogFileSize = Deprecated; no longer used. Use *[Debug]SysLogSize* instead.

Parameter Details:

DebugCategory defines which message categories to log. (See table below).

DebugLevel defines debug levels of messages to be logged. (See table below).

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

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 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]

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 PowerSCADA Expert 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

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, 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 scalability 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 229](#).) 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 229](#).) This means that, if some real-time tags are more important than 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 graphics can be resized.

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 PowerSCADA Expert 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 PowerSCADA Expert from a client computer, and you see a licensing error, make these port settings. This will open both PowerSCADA Expert 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 IDC	page downloads for IDC Internet display server/client communications
80	Web server	project files for Web client
2073	CTAPI	CTAPI communications
2074	client	Cicode debugging
2084	Reports server	Reports server communications
2080	Alarm server	Alarm server communications
2085	Trends server	Trends server communications
2078	IO server	legacy I/O 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?

PowerSCADA Expert cannot provide feedback about whether a write to a device register is successful. If a device is capable of 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.

How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes?

PowerSCADA Expert normally optimises its packets for greatest performance. This optimisation can sometimes include invalid areas of memory in devices. These invalid areas can be specifically defined and excluded from optimisation packets created by PowerSCADA Expert. See *Advanced Tag Blocking Capabilities* in [Performance Tuning Parameters on page 229](#) 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
```

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 PowerSCADA Expert. The only solution is to uninstall both Excel and PowerSCADA Expert; then, reinstall first Excel and then PowerSCADA Expert.

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 PowerSCADA Expert. 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.

Why is my device in comms loss?

When you bring your system on line, and you find that PowerSCADA Expert has lost communications with a device, check the following:

- Verify that the physical connection is correct and secure.
- Verify the IP address.
- Verify the Modbus address.
- Check the statusRegister, statusRegistersCount, and statusRegisterType (see for details)

How do I set up select before operate?

For systems in which you can determine that a single user is selecting a device prior to sending an open/close command, you can add a "select before operate" button.

To do this:

1. Locate the Select Before Operate tag in the variable tags.
2. Append `\str` to the end of the tag name.
3. Change the data type to STRING.
4. Click Add.

This creates the SBOw tag for the IEC 61850 advanced control screen. For more information about advanced control, see:

[Set Up IEC 61850 Advanced Control on page 117](#)

[Perform IEC 61850 Advanced Control on page 167](#)

Index of Terms

.aes file

encryption 207

1

1.#R

value returned in instant trend 163

A

access rights

adding for users 39

changing for users 211

Windows integrated users 210

ACE969TP device type

used with DNP3 50

ACE969TP module

used with DNP3 50

additional sources of information 9

address

correcting missing or invalid 24

functional 25, 53

defined 248

advanced control

perform for IEC 61850 165

set up

IEC 61850 117

advanced tag block capabilities

configuration notes 234

invalid block tag definition syntax 233

tag blocking notes 236

write-only memory 235

write-only tags 236

AdvOneLine.csv 120

AdvOneLine.exe 120

stopping and starting 123

AdvOneLine.ini 120

AdvOneLine.ini settings

one-lines 121

AdvOneLineDebug.Bus.csv 120

AdvOneLineStatusLog.txt 120

alarm

adding an onboard tag 141

associated with waveforms 150

category 161

category number 161

changing pickup/dropout text 259

changing the severity 140

codes

unique identifier 36

colors 156

filtering

custom 160, 162

historical logging parameters 140

issues 158

level

setting in Profile Editor 40

onboard 138

pc-based 138

setting up in Profile Editor 42

priority 161

priority number 161

- setting up based on an enumeration 139
- states
 - startup parameters 241
- tags
 - adding 139
- alarm filters 160
- alarm format code definitions 68
- alarm log 156
 - column headings described 159
 - filtering 156, 160
 - freezing 156
 - logging data to disk 157
 - printing 158
 - problem with alarms not displaying 140, 158
 - sorting 155-156
- Alarm On/Alarm Off text 42
- alarm pages
 - adding to the project 139
- Alarms/Events page
 - equipment column 155
 - filtering alarms 161
 - remove/insert/move columns 156
 - viewing 155
- AlarmSummaryLength parameter
 - maximum number of entries in the Event Log 140-141
- analysis page 163
- and all REAL 70
- animation
 - loss of communication not filtering correct 155
 - not display correctly when genie dimensions wrong 125
- animation numbers (ANs)
 - used in objects 258
- Application Services
 - restart 203
 - setup 204
- Applications 206
- array
 - genie properties 117
- assigning tags to generic I/O points 28
- assumptions
 - about PowerSCADA Expert system integrators 1
 - before system integrator can set up system 1
- ATS
 - configuring 124
 - errors 133
 - information 132
- audible alarms
 - setup 142
- auto assign genie 128, 131
- auto configure genie 128, 131
- automation interface
 - "out of memory" message 102
 - adding multiple devices 102
 - required information 103
- B**
- back ground color
 - changing in projects 112

background colors

changing 167

backup

Citect project file 143

Profile Editor project file 56

base unit and conversion

adding and editing 22

deleting 23

basic readings summary 184

basic reports

Internet Explorer 9 emulation 171

set up delivery method and email address 207

before you use the Profile Wizard 87

block writes

using 85

BRCB

entering for IEC 61850 device 99

entering for IEC61850 device 99

breaker

configuring 125

errors 134

information 134

browser navigation buttons do not work 112

busbar

configuring 125

creating and configuring 260

errors 134

information 133

busbars

two with same line active 131

C

CacheRefreshTime parameter 262

calculating disk storage 148, 163

CallProvider

PLSProviders.ci 143

CASE

changing user access rights 211

category/subcategory

adding 30

deleting 30

editing 30

managing 254

changing alarm severity 140

circuit breaker status

using special tags to control 67

Citect platform connection 205

Citect.ini

used to customize a project 143

citect.ini file

setting up multiple monitors 202

Citect.ini settings

one-line 121, 123

clear cache 145

Live View 192

reporting 178

cluster

adding 88

CLUSTER name (%CLUSTER%)

changing 264

- cluster setup
 - in the Profile Wizard 92, 94, 96, 98
 - redundant I/O servers 90
- collapse/expand all
 - Import Reconciliation screen 62
- color
 - active
 - busbars 261
 - changing background in projects 112
 - changing for genies 112
- colors
 - assigning to one-lines 121
 - changing background 167
- comment field
 - used in translation 89
- comms loss
 - troubleshooting 267
- communications loss 155
 - troubleshooting 267
- compile and run 6
- Compile Errors window
 - viewing PowerSCADA Expert errors 113
- Complete Import button 58, 63
- composite device type
 - creating 46
- Computer Setup Wizard
 - running after first compile 6, 88
 - when adding a startup page 112
- configuration
 - one-line engine 206
- configuration tools
 - described 2
- configure
 - application services 203
- control format codes 69
- control inhibit 167
- controls
 - how they operate 25, 37
- conversions
 - to base units 21
- core service host
 - connection with Citect 205
- core services
 - configure 203
- CoreServiceHost
 - restart when changing runtime projects 169
- create menu item for LiveView page 193
- creating
 - CSV file 29
 - device profile 39
 - engineering template 19
 - genie 116
 - one-line on a graphics page 112
 - PowerSCADA Expert project 110
 - project 50
- CSV file
 - creating, viewing, printing 18, 29
- CTZ file
 - backup file 143

- custom
 - control format codes 69
 - filters
 - alarms 160, 162
 - genie
 - creating 117
 - icon 17
 - reset
 - adding 36
 - symbol
 - creating 117
 - tag
 - deleting 34
 - deleting when associated with a device type 34
 - editing 33
 - custom tag
 - adding 32
 - customer support contact information 9
 - customize tag names
 - sub-tab 50
 - customizing a project with Citect.ini 143
 - customizing tag names 51, 54
- D**
- data concentrator
 - setup 48
 - data points
 - defining individually in an address 25
 - data replication parameter 240
 - datasets
 - adding/editing 44
 - DBF Add-In toolbar will not install 267
 - deadbands 148
 - debugging
 - INI parameters to use 261
 - define
 - device types and tags 23
 - enumeration 66
 - demand current summary 189
 - demand voltage summary 189
 - demo mode
 - using 202
 - design-time environment
 - described 110
 - device
 - adding multiple 256
 - adding to the PowerSCADA Expert project 92
 - name
 - requirements 255
 - popup
 - overlapping fields 264
 - profile
 - adding 52
 - creating 39
 - defined 6
 - deleting 43
 - editing 42
 - removing from a PowerSCADA Expert project 101
 - tags
 - defining 23

- type
 - adding, editing, deleting 26
 - defined 5
 - defining 23
 - deleting 28
 - editing 27
 - locking 17
 - unlocking 18
 - device communications issues
 - troubleshooting 255
 - device profile
 - add as default 53
 - device type
 - category/subcategory
 - adding 30
 - deleting 30
 - editing 30
 - composite 46
 - drivers
 - INI parameters 146
 - drivers and protocols
 - list of supported 3
 - name
 - limitations 26
 - Device Wizard
 - before you use it 87
 - devices in the PowerSCADA Expert project
 - editing 100
 - diagnostics (Application Configuration Utility 204
 - disabling "tag not exist" prompt. 113
 - disk storage calculations 148, 163
 - display client
 - basic report viewing 171
 - display projects button
 - adding a project in the Profile Editor 6, 52
 - DNP3-TCP device
 - adding 96
 - DNP3 protocol support 50
 - DNP3_serial
 - adding to PowerSCADA Expert project 94
 - documentation in this release 9
 - driver parameters
 - editing 4
 - selecting for a profile 47
 - setting up in the Profile Editor 42
 - drivers
 - choosing for device types 27
 - general PowerSCADA Expert 222
 - generic power device 3
 - how they work 3
 - subscription expirations 4
 - subscription types 3
 - duplicate devices
 - defined 4
- E**
- EcoStruxure Web Services (EWS)
 - configuration 204
 - avoid timeouts 205
 - introduction 169

- email
 - configure to send basic reports 173
 - sending reports 174
- encryption
 - aes file for saving INI file to a different computer 207
 - password 119, 122
- energy readings 185
- energy summary 185
- engineering template
 - creating 19
- engineering units
 - adding 22
 - exceptions 21
 - adding from the Project tab 53
 - deleting 23
 - editing 23
 - multipliers and offsets 21-22
- English.dbf
 - used in translation 93, 95, 97, 99
- English.dbf file
 - used for translation 93, 95, 97, 99
- entering copy for translation 91, 93, 95, 97, 99
- enumeration
 - defining 66
 - setting up alarm based on 139
 - status enumeration logic code 75
- equipment column
 - alarms/events pages 155
- equipment name
 - adding via Automation Interface 103
 - adding via Profile Wizard 93, 95, 97-98
 - using when creating a one-line 113
- equipment pop-up page 164
- Equipment.Profiles
 - file storage location 55
 - viewing 51
- errors
 - ATS 133
 - breaker 134
 - busbar 134
 - meter 135
 - sources 136
 - switch 133-134
 - transformer 137
- errors and warnings
 - correcting in the graphics page 113
 - one-line 130
- event filters 160
- event log 156
 - acknowledge, silence, print 157
 - logging data to disk 157
 - problem with alarms not displaying 140
- event notification service
 - Application Configuration Utility 205
- events
 - historical logging 140
 - storage limitations 140
- Example Automation.xls
 - using to export multiple devices 102
- Excel DBFAdd-In toolbar will not install 267

expanding PowerSCADA Expert screens

F2 key 89

export

PowerSCADA Export 55

Profile Editor (backup) 56

project files from Profile Editor 54

SCL files 57

F

F2 key

use to expand PowerSCADA Export
screens 89

use to expand screens 89

FAQs 254

"out of memory" message in Automation Inter-
face 266

#COM error for circuit breaker status 266

adding more than one device at a time 256

adding/removing devices in Profile Wizard 254

auditing user logins/logouts in the Event
Log 265

browser navigation buttons do not work 260

can I change %CLUSTER% name 264

changing default pickup/dropout text 259

creating and configuring busbars 260

creating device profiles without PowerLogic
drivers 254

creating users in Runtime 256

device name requirements 255

device type vs device profile 254

device writing to its registers 265

Excel DBF Add-In toolbar will not install 267

How do I troubleshoot device communications
issues? 255

how to use MODBUS communications
methods 256

if tag is defined, is it polled 263

INI parameters used for debugging 261

licensing error at web client 264

managing categories and subcategories 254

managing projects in PowerSCADA
Export 257

naming conventions for servers/clusters 256

overlapping fields in device popup 264

performance tuning parameters 262

preventing reads/writes to invalid areas of
memory 265

synchronize new PC with master PC 254

what are genies 258

what can be set up in logging and
archiving 260

what can I modify during runtime? 260

working with templates 258

FileFormat parameter

historical logging of alarms and events 141,
157

filter

import data 61-62

filter genie states 129, 159

filter the alarm and event logs 160

filters

real time and alarm 33

First dbf record

error 267

force edit 63

format codes

alarm definitions 68

- control definitions 69
 - custom control 69
 - definitions 67
 - predefined control 69
 - real-time definitions 67
 - reset 69
 - frm file
 - associated with new genie 258
 - FRM file
 - formatting genie appearance 117
 - functional address 25, 53
 - definition 248
 - edit 31
 - functional constraints
 - IEC 61850 59
 - fundamental phasor readings 185
- G**
- G3200 setup 48
 - generic I/O points
 - assigning tags 28
 - generic MODBUS drivers
 - included in system 3
 - genie
 - adding to a drawing 112
 - auto assign 128, 131
 - auto configure 128, 131
 - busbar 125
 - configuration
 - viewing 128
 - creating and editing 116
 - grouping 129
 - icon 112
 - in Graphics page 258
 - libraries
 - ITEM1 220
 - PLS_ALARM 212
 - PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2 212
 - PLS_ANSI_CB_1 / PLS_ANSI_CB_2 214
 - PLS_ANSI_SW_1 / PLS_ANSI_SW_2 215
 - PLS_DISPLAY 215
 - PLS_GEN_BASE_1 / PLS_GEN_BASE_2 216
 - PLS_GEN_CMD_1 / PLS_GEN_CMD_2 217
 - PLS_GEN_EQ_1 / PLS_GEN_EQ_2 217
 - PLS_IEC_BASE_1 / PLS_IEC_BASE_2 218
 - PLS_IEC_CB_1 / PLS_IEC_CB_2 218
 - PLS_IEC_SW_1 / PLS_IEC_SW_2 219
 - PLS_METER 219
 - meter 126
 - repairing 131
 - resizing 125
 - source 126
 - state filters 129
 - switch/circuit breaker 125
 - transformer 127
 - genie color
 - changing in project pages 112
 - GenieConfiguration.xml file
 - one-line 127

genies

PowerSCADA Expert 124

GetProviderResult

PLSProviders.ci 145

GetProviderStatus

PLSProviders.ci 144

getting more information 9

graphics library parameters 241

graphics page

adding in PowerSCADA Export 111

setting as startup page 112

Graphics runtime environment

editing appearance 115

group by type or page 129

guest user

single sign on 208

H

harmonic apparent power flows 187

harmonic reactive power flows 188

harmonic real power flows 188

help files 9

historical parameters of alarms

setup 140

historical trends

viewing 163

viewing from a one-line 166

HostName

ini parameter 242

I

I/O server

adding before using the Equipment Wizard 88

ICD file

editing 57

icons

locked and custom 17

IEC

import filter for ICD files 59

IEC 61850

adding device to PLS project 98

advanced control setup 117

datasets

adding/editing 44

functional constraints 59

general workflow 8

perform advanced control 167

re-match 62

report control blocks 59

adding/editing 45

tag construction 65

tags

logic codes described 69

IED

defined 248

IF logic

Live View 196

ignored devices and topics 148

IID

defined 248

import

- files into Profile Editor 57
- filter data 61
- ignore 63
- match option 63
- project files into Profile Editor 54
- removing import templates 65
- SCL files 59
- set to unmatched 63
- template
 - applying 58
 - using 64

import filter

- IEC 61850 files 59

import Import reconciliation screen 60**incremental reactive energy summary 186****incremental real energy summary 187****INI parameters 221**

- for historical reports 170
- for supported device type drivers 146
- to use in debugging 261

instant trend

- pens stop updating 163
- viewing 163

interlock check

- IEC 61850 devices 167

internationalization

- real-time tables 199

Internet Explorer 9 emulation

- reporting 171

intervals

- trend tags 47

invalid or missing tag address

- correction 24

IP address

- entering in Example Automation.xls 103, 107

Is Device Tag

- reporting for composite devices 41

J**JBus**

- adding 3, 27

L**launching the profile editor 17****licensing error at web client 264****LiveView**

- basic readings summary 184
- conditional formulas 196
- create menu item 193
- create new template 194
- data format 195
- delete 199
- demand current summary 189
- demand voltage summary 189
- does not launch (Matrikon Explorer) 183
- duplicate 198
- edit predefined template 192
- energy readings 185
- energy summary 185
- formatting 194
- formulas 196

fundamental phasor readings 185
harmonic apparent power flows 187
harmonic reactive power flows 188
harmonic real power flows 188
incremental reactive energy summary 186
incremental real energy summary 187
internationalization 199
introduction 183
lock setup 192
minimum/maximum readings 197
modify 198
placeholders 191, 195
power flow summary 184
save a template 193
save a view 191
setup 192
tables list 184
templates 190
THD current summary 185
THD voltage summary 186
thresholds 197
update device list 190
update intervals 191
update list 190
update now 191
uptime summary 186
URL link to launch 190
view area 193
viewer window 189
views 190

Where's my device? 191
LiveView Template Editor 183
locked device type
 cannot delete 28
 unlocking 18
locked icon 17
locking a device type 17
lockout/tagout 154
 enable 114
logging level 204
logic codes
 definitions 69
loopback address
 used for network addresses 88-89

M

main menu options 18
manuals 9
Matrikon
 LiveView and reporting do not launch 170, 183
memory device
 one-lines 120
menu
 main options 18
Menu Configuration
 used to edit Graphics page menu
 information 6, 115
Menu Configuration tool 114
metadata/fields.xml
 use to edit alarm filters 161
meter
 configuring 126

- errors 135
- information 135
- Micrologic
 - genie in Graphics Builder 219
 - MicrologicV INI settings 240
- Microsoft Excel
 - using to add devices to the database 102
- migration utility 10
- MODBUS communications methods 256
- multi-device reports 179
- multiple monitor support 202

N

- naming ports 100
- navigation arrows
 - runtime environment 154
- network address
 - adding 89
- network address and servers
 - adding before using Profile Wizard 88

NetworkTagsDev I/O device

- adding 90
- normally closed
 - control tags 25

O

- OLE automation interface
 - add and remove devices 103
- onboard alarm tag
 - adding 141
- onboard alarms 138

- one-line
 - adding to a graphics page 112
 - coloring 121
 - communication errors 130
 - PowerSCADA Expert
 - described 120
 - repairing 131
- one-line animation file locations 120
- One-Line Configuration Utility 128
- one-line device 120
- one-line engine
 - configuration 206
- one-line flowchart 118
- one-line memory device 120
- out of memory message
 - when running Automation Interface 102
- override
 - tag name 41, 47
- override tag status 166

P

- pack
 - after deleting a cluster 89
 - using to delete data 118
- Pagemenu.dbf
 - editing 115
- parameters
 - advanced tag block capabilities 232
 - alarm and event state 241
 - AlarmFormat 160, 162
 - AlarmSummaryLength 140-141

- AlarmUseConfigLimits 139
- Bandwidth Allocation (port level) 229
- BandwidthAllocation (device level) 230
- cacheRefreshTime 226
- CCM (Micrologic) 239
- Citect INI 221
- ClientAutoLoginMode
 - used to set up Windows integrated user 211
- CommandsBandwidth 229
- data replication 240
- database described 221
- driver
 - editing 4
 - selecting for a profile 47
- enableScatteredReads 230
- EventBandwidth 229
- EventIdle 239
- EventTable 238
- general PowerSCADA Expert drivers 222
- graphics library 241
- HighScanRate 232
- initCacheTimeout 225
- initUnitCheckTime 225
- kernelStatisticUpdateRate 222
- LowScanRate 232
- maxBlockSize 231
- Micrologic 239
- MicrologicV INI settings 240
- minBlockSize 231
- ModbusBase 227
- module-specific packet blocking optimisation settings 239
- packet blocking optimization 230
- percentBlockFill 230
- performance tuning 229, 262
- PLSEventLog 141
- PLSEventLogFileFormat 157
- RegMode 228
- retry 223
- retryexception 224
- retrytimeout 223
- Sepam Event Reading 238
- standbyCheckTime 224
- standbyRefreshRate 224
- statusRegister 226-227
- statusregistertype 227
- statusUnitCheckTime 225
- Tag Scan Rate 232
- time synchronization 147
- time zone settings 146
- TimeBandwidth 229
- timeout 223
- TimeSync 226
- timeZone 228
- usewritemultiregistersonly 222
- watchtime 222
- waveform 237
- WaveformBandwidth 229
- WaveformCheckTime 238
- WaveformMatchMargin 238
- WaveformsDisable 237

- WaveformZone 238
- password encryption 119, 122
- pc-based alarms 138
 - setting up 42
- performance tuning parameters 229, 262
- pickup/dropout text
 - changing for alarms 259
- platform refresh 145
 - LiveView 192
 - reporting 178
- PLS export 55
- PLS_ClearCache 145
- PLS_Example project 7, 110
- PLS_PPlatformRefresh 146
- PLS_ReportDsp()
 - Cicode function
 - used in report configuration 169
- PLS_ReportPage
 - used in report definition 169
- PLSAnimBusbar()
 - Cicode function for custom genie animation 117
- PLSEventLog parameter
 - allowable values 157
 - historical logging of events 141
- PLSProviders.ci module 143
- PM5000
 - device profiles for the Profile Wizard 93, 95
- PM5000S 93, 95
- PM5000S1 93, 95
- PMCU
 - adding onboard alarms and waveform captures 140
 - defined 249
- polling rules for tags 263
- port renaming
 - serial 94
- ports
 - naming 100
 - renaming
 - serial 95
 - TCP 93, 97
- power flow summary 184
- PowerSCADA Expert
 - backup 143
 - described 2
 - naming conventions 256
 - project
 - creating 110
 - uninstall and reinstall 5
- PowerSCADA Expert one-lines
 - described 120
 - error logging 130
 - genie introduction 124
 - GenieConfigureation.xml file 127
 - introduction 128
 - SupportedGenies.xml file 127
- predefined control format codes 69
- predefined project
 - PLS_Example 110

- printing
 - CSV file 29
- printing the alarm log 158
- privileges
 - user
 - PowerSCADA Expert 209
- profile editor
 - export 56
 - launching 17
 - overview 13
- Profile Editor
 - importing files into 57
- Profile Editor.exe.config
 - using to change trend tag scan intervals 148
- Profile Wizard
 - adding a device 87
 - adding a device to a project 92
 - before you use it 87
 - editing device information in a PowerSCADA Expert project 100
 - removing a device 101
- profiles
 - adding, editing, deleting 41
 - enabling waveforms 39
 - file storage location 55
 - name limitations 41
 - viewing 39
- project
 - adding 52
 - cannot add 267
 - creating in PowerSCADA Expert database 55
 - deleting 53
 - editing 53
 - exporting (to Citect from Profile Editor) 55
 - managing in PowerSCADA Expert 257
 - name limitations 52
 - problems when renaming in Citect 110
 - reusing 111
 - setting up 50
- project parameters
 - sub-tab 50
 - using to add information for imports 51
- Project Parameters sub-tab 54
- project XML files
 - storage location 55
- PSEHostName
 - ini parameter for SSO 242
- Q**
- Quantum PLC
 - genie in Graphics Builder 220
 - time-stamped events 244
- quick start 5
- R**
- rack position
 - displaying incorrectly 125
- Rapid Access Label
 - configure the report 172
 - create and use 200
 - label stock 201
 - view LiveView tables 190
- re-match IEC 61850 items 62

- read logic codes 70
- read/write logic codes 85
- real-time format codes definitions 67
- real-time lists
 - viewing from genies 166
- redundant I/O servers
 - cluster setup 90
- refreshing tags in the Project tab 51, 55
- RemoteCallHandlerCluster
 - ini 243
- RemoteCallHandlerServer
 - ini 242
- repair
 - one lines 131
 - project 129
 - upgrade project 129
- report configuration
 - saving 172
- report control blocks
 - IEC 61850 59
- reporting basic
 - application tasks 171
- ReportMailer utility
 - sending email reports 175
- reports
 - adding to runtime menu 170
 - basic
 - accessing, creating, modifying, removing 171
 - configure email to send 173
 - editing appearance 178
 - export 177
 - exporting, printing, saving 176
 - localization 182
 - set up a display client for report viewing 171
 - set up delivery method and email address 207
 - URL routing 182
 - do not launch (Matrikon Explorer) 170
 - enable Windows Authentication 181
 - feature described 169
 - INI parameters 170
 - introduction 178
 - multi device usage 179
 - PLS_ReportDsp() 169
 - PLS_ReportPage 169
 - sending via email 174
 - setting up in the runtime menu 170
 - single device usage 178
 - SMTP server and from address 173
 - tabular 180
 - trend 180
 - using ReportMailer 175
- reserved names
 - do not use in projects 250
- reset format codes 69
- resets
 - adding custom 36
 - how they operate 25, 37
- restart services 203

- restore
 - project file 142
- resx files
 - localization 182
- reusing a project file 56, 111
- roles
 - adding user 210
- runtime environment
 - described 3
 - launching and closing 153
 - viewing 153
- S**
- SBOw tag
 - creating 267
- SCADA
 - defined 1
- scan intervals
 - trend tags 147
- scattered reads
 - using enableScatteredReads parameter 230
- SCL
 - defined 251
 - export 57
- SCL files
 - importing 59
- screen resolution
 - Profile Editor 18
- security viewer 161
 - filter 162
- select before operate
 - setup 267
- Select Devices
 - LiveView 190
- Selected Device Profiles
 - sub-tab 50
- selecting 21
- Sepam
 - genie in Graphics Builder 220
- sequence of events
 - alarm/event logs 160
- SER (sequence of events recorder) 3, 46, 159, 161
- serial device
 - adding to PowerSCADA Expert project 94
- servers
 - adding 89
 - adding to PowerSCADA Expert 88
 - described 88
- servers/clusters
 - naming conventions 256
- service inventory 204
- setpoints and delays
 - entering 139
 - entering from the runtime environment 165
- setting up
 - alarm based on an enumeration 139
 - audible alarms 142
 - device type categories 29
 - normally closed/normally open controls 25

- severity
 - changing for alarms 140
 - show by type or page 129
 - sim source
 - transformer top and bottom 137
 - single device reports 178
 - single sign on 208
 - guest user 208
 - SMTP server
 - for reports 173
 - sorting the alarm log 40, 155-156
 - source
 - configuring 126
 - errors 136
 - information 136
 - start and stop AdvOneLine 123
 - startup page
 - setting 112
 - state filters 129
 - storage limitations
 - calculating disk space 148, 163
 - subscriptions
 - drivers 3
 - support
 - technical 9
 - SupportedGenies.xml file
 - PowerSCADA Expert one-lines 127
 - switch
 - configuring 125
 - errors 134
 - information 132, 134
 - symbol
 - creating custom for custom genie 117
 - synchro check
 - IEC 61850 devices 167
 - set up
 - IEC 61850 117
 - synchronize new PC with master PC 254
 - system creation steps 5
- T**
- tabular usage reports 180
 - tag
 - custom
 - adding 32
 - customizing the name 51
 - IEC 61850 described 65
 - refreshing in the Project tab 51
 - tag-polling rules 263
 - tag addresses
 - adding 38
 - control 37
 - correcting missing or invalid 24
 - editing 34, 38
 - generic 39
 - onboard alarm 36
 - real-time 34
 - reset 36
 - tag count
 - driver subscriptions 3

- tag definitions
 - printing the CSV file 29
- tag names
 - customizing 54
- tag override 41, 47
- tag status
 - overriding 166
- tag types 65
- tags
 - assigning to generic I/O points 28
 - scanning intervals for trend tags 47
- TCP device
 - adding to PowerSCADA Expert project 94, 98-99
 - adding to project 92
- technical support 9
- Template Editor
 - LiveView 183
 - lock 192
- templates
 - creating 19
 - deleting 21
 - imports 64
 - working with 258
- THD current summary 185
- THD voltage summary 186
- The_Profile_Wizard 87
- time-stamped events
 - Quantum PLC 244
- time quality
 - alarm/event logs 160
- Time Sync
 - filter for alarm or event log 161
- time synchronization parameter 147
- time zone setting parameter 146
- transfer switch
 - configuring 124
- transformer
 - configuring 127
 - errors 137
 - information 137
 - top and bottom 137
 - voltage-level and pass-through 121, 127, 138
- translation
 - using description fields 91
- trend tag intervals 163
 - changing in the Profile Editor 43
- trend tag scan intervals 47, 147
- trend usage reports 180
- trending (Analysis Page)
 - described 163
- trends
 - maximum number of pens 163
 - setting up in Profile Editor 42-43
 - viewing from within a one-line 166
- typical workflow
 - creating a graphics page 109
 - creating a project in Profile Editor 51
 - creating a system 5
 - Profile Editor 13

U

- uninstalling and reinstalling PowerSCADA Expert 5
- unique ID
 - determining for onboard alarms 36, 141
- units (engineering) and conversions
 - creating or editing a units template 19, 21
 - deleting 23
 - deleting a unit template 21
 - editing 23
 - exceptions 21
- uptime summary 186
- URCB
 - entering for IEC 61850 device 99
- URL
 - visit to send reports via email 174
- URL link
 - launch Live View Viewer 190
- URL reporting for basic reports 182
- UsePLSFilter
 - alarm/event filter parameter 160, 229
- user access
 - adding and modifying 209
 - changing rights 211
 - Windows integrated 210
- useraccess
 - default settings 209
- users
 - creating in Runtime environment 256
- using block writes 85

V

- value type
 - controlling real-time display 165
- Variable tags not found
 - disabling the prompt 113

W

- waveform
 - why can't I acquire? 266
- waveform parameters 237
- waveforms
 - enabling in Create Device Profiles tab 39, 52
 - enabling onboard 140
 - linking to alarms 150
 - management 150
 - special digital tags 151
 - storage rules 150
 - viewing from the Alarm Log 157
 - viewing from the runtime environment 157
 - viewing in the runtime environment 165
- WaveWeb waveform viewer 157
 - PDF location 165
- web.config
 - securing a URL for sending reports via email 174
 - SMTP server used when sending email 173
 - specify Windows Authentication, adding roles for LiveView 200
 - specify Windows Authentication, adding roles for Reporting 181
 - specify Windows Authentication, adding roles for Live View 200
- Where's my device? 191

Windows Authentication

enable for Live View 199

enable for reports 181

Windows integrated user accounts 210

Windows security

using to remove user access rights 209

write logic codes 84

writes (logic codes) 85

X

XML files

storage for project files 55

Z

zOL

memory device for one-lines 120

