

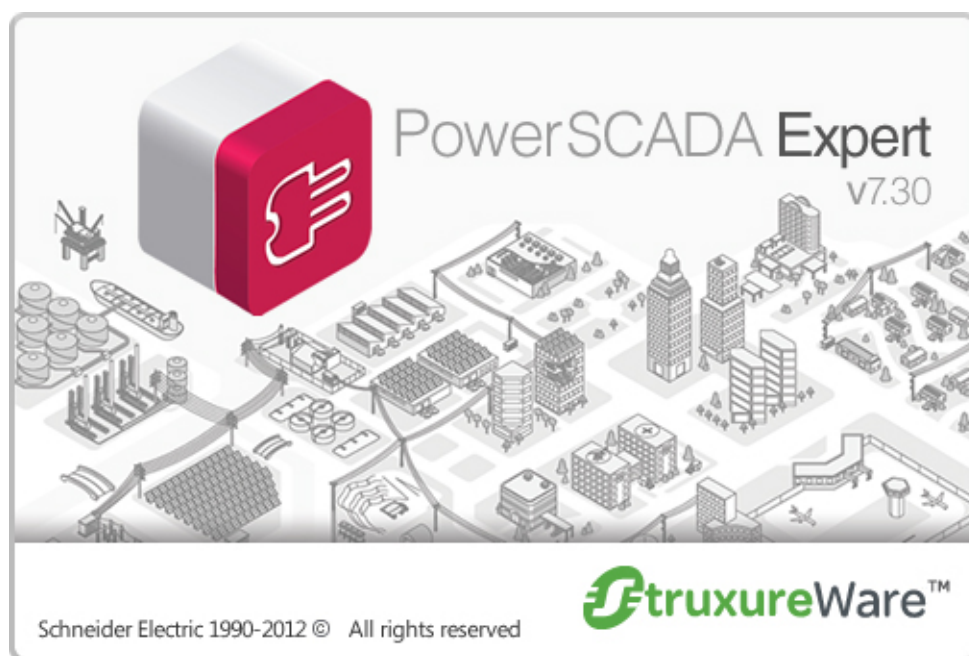
PowerSCADA Expert

System integrators manual

Version 7.30

63220-100-200F1

01/2013



Notices

ION, ION Enterprise, Modbus, Modicon, PowerLogic, Schneider Electric, and System Manager are either trademarks or registered trademarks of Schneider Electric in France, the USA, and other countries. All other trademarks are property of their respective owners.

Contents

Contents	1
Introduction	10
PowerSCADA Expert—A Complete Solution	11
Assumptions	11
What is SCADA?	11
What is PowerSCADA Expert?	11
Prepare for PowerSCADA Expert	11
Components of a Project	12
Configuration Tools	12
Runtime Environment	12
Supported Device Types and Protocols	12
How Do Drivers Work?	13
Two Subscription Types	13
Subscription Expirations	13
Edit Driver Parameters	13
Problems with Duplicate Devices using PowerLogic Drivers	13
System Requirements and Installation Instructions	15
Uninstall and Reinstall PowerSCADA Expert	15
Quick Start: Typical Workflow	16
IEC 61850 System Setup Workflow	19
Get More Information	20
Manuals	20
Help files	20
Customer support	20
Use the Migration Utility	21
Use the Migration Utility	22
The Profile Editor	23
Overview of the Profile Editor	25
Typical Workflow Illustration	26
Workflow Overview	26
Create/Edit Device Type	27
Create/Edit Device Profile	28
Create/Edit Unit Templates	29
Launch the Profile Editor	30
Locked and Custom Icons	30
Set the Screen Resolution	30
Main Menu Options	31
Add Engineering Unit Templates, Units, and Conversions	32
Set Up Engineering Templates and Select Conversions	32
Apply Conversions	34
Delete a Template	34
Add or Edit a Base Engineering Unit or Conversion	34
Edit a Base Engineering Unit or Conversion	35
Delete a Base Engineering Unit or Conversion	35
Define Device Types and Tags	36
Device Type Screens and Workflow	36
Use the Define Device Type Tags Tab	36
Add, Edit, or Delete a Device Type	38
Edit a Device Type	40
Delete a Device Type	40
Assign Tags to Generic I/O Points	41
Print the .CSV File	41
Set Up Device Type Categories	42
Add a Category or Subcategory	42
Edit a Category/Subcategory Name	42
Delete a Category or Subcategory	43
Edit Functional Addresses	43

Add Custom Tags and Tag Addresses	44
Set Up Custom Tags	44
Edit a Custom Tag	46
Delete a Custom Tag	46
Edit Tag Addresses	46
Real-Time Tag Addresses	47
Onboard Alarm Tag Addresses	48
Reset Tag Addresses	48
Control Tag Addresses	49
Edit Address Information	50
Add a New Tag Address	50
Edit Generic Tag Addresses	50
Create Device Profiles	51
Enable Waveforms	51
View Device Profiles	51
Add Edit or Delete Device Profile	52
Add a Profile	52
Edit a Profile	54
Delete a Profile	54
Select Trend Intervals	55
Set Up Trend Intervals	55
To add a trend interval:	55
To edit a trend interval:	55
To delete a trend interval:	55
Edit IEC 61850 Datasets	55
Create a New DataSet	56
Create a DataSet from an Existing DataSet	56
Copy a DataSet to a Device Type	56
Edit and Delete DataSets	56
Edit IEC 61850 Report Control Blocks	57
Create a New Report Control Block	57
Create a Report Control Block from an Existing Report Control Block	57
Copy a Report Control Block to a Device Type	57
Edit and Delete Report Control Blocks	58
Create a Composite Device Type	58
G3200 Device Setup	60
For use with multiple devices	60
In the Profile Editor	60
In SFT850	60
In PowerSCADA Expert	60
For use with a single device	60
In the Profile Editor	60
In SFT850	61
In PowerSCADA Expert	61
IEC 60870-5-104 Protocol Support	61
DNP3 Protocol Support	61
Set Up Projects in the Profile Editor	62
Project Screens and Workflow	62
Typical Workflow	62
The Set Up Projects Tab	63
Add, Edit, or Delete a Project	63
Adding a Project	64
Edit a Project	65
Delete a Project	65
Edit and Delete Information in a Project	65
Customize Tag Names	65
Add Project Parameters	65
Import and Export Project Files	66
Export a Project	66
Before you export	66
To export a Profile Editor project to the PowerSCADA Expert project:	66

To Move Files if the Profile Editor is not with the Server (Reuse of a Project)	67
Profile Editor Export	68
SCL Export	68
Exporting the File	69
Import Files into the Profile Editor	69
Import Reconciliation Screen (Search)	70
Edit the Item	71
Complete the Import of an Item	71
Import Reconciliation Screen (Merge-Match)	71
Import SCL Files	73
Using Import Templates	73
Creating a New Template During Import	73
Applying a Template During Import	74
Deleting a Template	74
Tag Types	75
IEC61850 Tag Construction	75
Define an Enumeration	76
Use Special Tags to Control Circuit Breaker Status	76
Format Code Definitions	77
Real-Time Format Code Definitions	77
Alarm Format Code Definitions	78
Control Format Code Definitions Rules of Operation	78
Predefined Control Format Codes	78
Predefined Reset Format Codes	79
Custom Control and Reset Format Codes	79
Logic Codes	79
Block Writes	86
The Profile Wizard	88
Before You Use the Profile Wizard	89
Add the Cluster, Network Address, and Servers	89
When You Set Up More than two I/O Servers per Cluster	91
Enter Copy for Translation	92
Using the Wizard to Add and Remove Devices	93
Add a Device to a Project	93
Add a TCP Device to a Project	93
Add the TCP Device	94
Add a Serial Device to a Project	95
Adding the Serial Device	95
Add a DNP3_TCP Device	97
Add the DNP3_TCP Device	97
Add an IEC61850 Device to a Project	99
Add the IEC 61850 Device	99
LDName	100
BRCBs and URCBs	100
Port Names	100
Edit Devices in a PowerSCADA Expert Project	101
Edit a Profile and Add it Back to the Project	101
Edit a Device in PowerSCADA Expert Only	102
Add Device Data in PowerSCADA Expert Only	102
Remove a Device From the Project	102
Add Multiple Devices through the Automation Interface	103
Use the OLE Automation Interface to Add and Remove Devices	105
Examples	105
C#	105
Excel VBA	106
Command Line Interface	106
Interface Workflow	106
Prerequisite information	107
Interface	108

Graphics: the Design Time Environment	110
Typical Workflow	112
The PowerSCADA Expert Pages	113
Create a Project	114
Reuse Projects Created in the Profile Editor	114
Before You Begin	114
Add a New Graphics Page	115
Change the Background Colour of Pages	115
Change the Genie Colour in Project Pages	115
Set a New Page as the Project Startup Page	116
Create a One-Line on a Graphics Page	116
Use Menu Configuration to Edit Pagemenu.dbf (Change the Graphics Page Appearance)	117
Delete Information from PowerSCADA Expert	119
Use PowerSCADA Expert One-Lines	120
One-Line Flowchart	120
What Are PowerSCADA Expert One-Lines?	121
Where are the files located?	122
One-Line Device (zOL)	122
One-Line Colours	122
Assigning Colours	123
Add INI Settings to AdvOneLine.ini and Citect.ini	123
AdvOneLine.ini Settings	123
Citect.ini Settings	124
Start and Stop One-Lines	124
Work with Genies	124
Genie Type Descriptions	125
Configure an Automatic Transfer Switch (ATS)	125
Transfer Switch Information - Left/Right/Bottom	125
Display Information	125
Configure a Busbar	125
Configure a Circuit Breaker or Switch	126
Configure a Meter	126
Configure a Source	127
Configure a Transformer	127
Create New Genies	128
To create a copy of a genie:	128
To create a custom symbol for a custom genie:	129
GenieConfiguration.xml File	129
SupportedGenies.xml File	129
Work in the Configuration Utility	129
One-Line Configuration Utility	130
Modify AdvOneLine.csv	130
Review Genie Configurations	130
One-Line Errors and Warnings	131
Communication Errors	131
Error Logging	132
Perform Repairs for One-Lines	132
Automatic Transfer Switch (ATS) Information	134
ATS Information	134
ATS Errors	134
Busbar Information	135
Busbar Errors	135
Breaker and Switch Information	135
Breaker and Switch Errors	136
Meter Information	136
Meter Errors and Warnings	137
Source Information	137
Source Errors and Warnings	137
Transformer Information	138
Transformer Errors	138

Work with Alarms	140
Alarms Overview	140
PC-Based Alarms	140
Onboard Alarms	140
Set Up Alarm Pages	140
Add Setpoints and Delays	140
Set Up an Alarm Based on an Enumeration	141
Change an Alarm Severity	141
Enable Waveforms for Onboard Alarms	141
Set Parameters for Event Log Length and Historical Logging of Events	142
Event Storage: [Alarm]SummaryLength Parameter	142
Event Logging: [PLSEventLog] Parameter	142
Add an Onboard Alarm Tag	143
Set Up Audible Alarms	143
Restore and Back Up Projects	144
Restore a Project	144
PowerSCADA Expert Backup	144
Use Cicode to Customize a Project	145
Customize Default Behaviours	146
Time Zone Settings	146
Time Synchronization	146
Trend Tag Scan Intervals	147
Disk Storage Calculation for Trends	148
Waveform Management	149
Waveform Storage	150
Waveform Database and Special Waveform Tags	151
Graphics: the Runtime Environment	152
Launch and Close the Runtime Environment	153
View the Graphics Page in the Runtime Environment	154
Log On to the Runtime Environment	154
View the Interface	154
Communications Loss	155
Viewing the Alarms/Events Page	155
Equipment Column	155
Filter Information	155
Remove/Insert/Move Columns	155
Sort by Column	156
The Event Log	156
The Alarm Log	156
Unacknowledged Alarms and Disabled Alarms	156
Alarm and Events Logging	156
Acknowledge, Silence and Print	157
When Alarms do not Display Correctly	157
Event/Alarm Log Columns Table	158
Alarm/Event Filter Form	159
Use the Analysis Page	160
Use the Equipment Pop-Up Page	161
View Waveforms	162
Enter Setpoints for Alarms	164
View Real-Time Trends	164
View Lists of Real-Time Information for the Genie	165
Override Tag Status	165
Use Reports	166
Set up the Runtime Environment for Reporting	166
Create the Menu Items for Report Page	166
Add the Following INI Parameters	166
Set Up the CtAPI Connection for Reporting	167
Set Up a Display Client for Report Viewing	167
Tasks in the Reporting Application	167
Create and View Reports	168

Run a New Report	168
Run a Report and Save its Configuration	168
View a Report Using a Saved Configuration	169
Modify and View a Report Using a Saved Configuration	169
Remove a Saved Configuration	170
Configure Email Settings to Send Reports	170
SMTP Server and From Address	170
Email Lists	170
Email Body	171
Send PowerSCADA Expert Reports via Email	171
Scheduling Reports	173
Read, Export, Print, and Edit Reports	173
Toolbar Options	173
Export a Report	174
Edit the Report Appearance	174
PowerSCADA Expert Reports	174
Single Device Usage Reports	175
Multi Device Usage Reports	175
Tabular Reports	176
Trend Reports	177
Enable Windows Authentication for Reporting	177
URL Routing for Reports	178
Windows 2008 R2 and Windows 7	178
Windows XP	178
Report Localization	179
Multi-Monitor Support	180
The Demo Mode	181
Assign and Control User Privileges	182
Default User Access Settings (Privileges)	183
Add and Modify User Accounts	184
Use Windows Integrated Users	184
Change Access Rights	186
Appendix 1: Default Genie Library	187
PLS_ALARM	187
PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2	187
PLS_ANSI_CB_1 / PLS_ANSI_CB_2	188
PLS_ANSI_SW_1 / PLS_ANSI_SW_2	190
PLS_DISPLAY	190
PLS_GEN_BASE_1 / PLS_GEN_BASE_2	190
PLS_GEN_CMD_1 / PLS_GEN_CMD_2	191
PLS_GEN_EQ_1 / PLS_GEN_EQ_2	192
PLS_IEC_BASE_1 / PLS_IEC_BASE_2	192
PLS_IEC_CB_1 / PLS_IEC_CB_2	193
PLS_IEC_SW_1 / PLS_IEC_SW_2	193
PLS_METER	194
ITEM1	194
Appendix 2: Citect INI Parameters	195
Parameters Database	195
General PowerSCADA Expert Parameters	197
watchtime	197
kernelStatisticUpdateRate	197
UseWriteMultiRegistersOnly	197
timeout	197
retry	198
RetryTimeout	198
RetryException	198
standbyRefreshRate	199
standbyCheckTime	199
statusUnitCheckTime	199
initUnitCheckTime	200

initCacheTimeout	200
cacheRefreshTime	200
StatusRegister	201
StatusRegistersCount	201
StatusRegisterType	201
ModbusBase	202
RegMode	202
timeZone	202
Alarm Parameters	204
UsePLSFilter	204
Performance Tuning Parameters	205
Bandwidth Allocation Parameters	205
Packet Blocking Optimisation Parameters	205
Advanced Tag Block Capabilities (Invalid Memory Access Blocks defined)	208
Invalid Block Tag Definition Syntax	209
Configuration Notes	209
Write-only Memory	210
Tag Blocking Notes	211
Write-only Tags	211
Waveform Parameters	213
WaveformsDisable	213
WaveformMatchMargin	213
WaveformCheckTime (PM, CM, and Sepam)	213
WaveformZone (Sepam)	214
Sepam Event Reading Parameters	215
EventTable	215
EventIdle	215
MicroLogic Modules Configuration Parameters	216
CCM	216
Module-Specific Packet Blocking Optimisation Settings	216
Data Replication Parameters	217
Database root folder path	217
Database root UNC path	217
Replication destination configuration	217
Graphics Library Parameters	218
Maximum number of entries that can be held in Event Log	218
Parameters for Alarm and Event States	218
Appendix 3: Additional INI Parameters	220
Quantum PLC Time-Stamped Events	220
Appendix 4: Glossary	221
address	221
alarm categorization	221
alarm text (On/Off)	221
alarm filters	221
alarm groups	221
alarm levels	221
alarm types	221
Automation Interface	221
bandwidth	221
baud rate	221
bitmask	222
Cicode	222
cluster	222
CommsMethod (communications method)	222
ComPort	222
composite device type	222
configuration environment	222
control	222
custom device type	222
custom tag	223

DataBits	223
data type	223
demo mode	223
design time environment	223
device category	223
device profile	223
device type	223
device type drivers	224
engineering unit templates	224
enumeration (used for the circuit breaker status)	224
format code	224
functional addressing	224
genie	224
ICD file	224
IEC tag name	224
IED	225
IID	225
logic code	225
metadata	225
multi-monitor support	225
onboard alarm	225
parity	225
PC-based alarms	225
PMCU	225
point (see SCADA tag)	226
polling priority	226
PowerSCADA Expert tag name library	226
Profile Editor	226
Profile Wizard	226
project	226
register scaling	226
resets	226
reserved names	227
runtime environment	227
SCADA tag (SCADA point)	227
SCL	227
StopBits	227
super-genie	227
tag	227
tag address	228
tag description	228
tag group	228
units	228
units template	229
user privileges (user access, user rights)	229
vector math	229
Appendix 5: Frequently Asked Questions (FAQs)	230
If I don't use PowerLogic drivers, how do I create device profiles?	230
The easiest way to do this is to:	231
Create a device type using a non-PowerLogic driver (like MODNET).	232
Can I install ION E or SMS on the same computer as PowerSCADA Expert?	232
How should we manage categories and subcategories?	232
When should I create a device type rather than device profile?	232
How do we synchronize a new PC with the master Profile Editor PC?	232
What do I do before I add or remove devices in the Profile Wizard?	232
What are the requirements for device names?	233
How do I troubleshoot device communications issues?	233
How do I use MODBUS communications methods?	234
How can I add more than one device at a time?	234
What are the naming conventions for servers and clusters?	234
How and when do I create users for the Runtime environment?	234

How do I manage projects in the Citect Explorer environment of PowerSCADA Expert?	235
On the Graphics page, what do I need to know about creating genies?	236
How do we customize existing templates?	236
How do I change the default pickup/dropout text for alarms?	237
What can I modify during runtime?	237
Why do the browser navigation buttons not work?	238
What can I set up in logging and archiving?	238
How do I create and configure busbars?	238
What INI parameters should I use for debugging?	239
How do I tune my system for best performance?	240
If a tag is configured, how is it polled in the device?	241
Device popup from a one-line: Why do the fields overlap?	241
Can I change the %CLUSTER% name in the Profile Wizard?	242
When viewing PowerSCADA Expert from a web client, I see a licensing error. How do I fix it?	242
A device can prevent writes to its registers: how do I ensure that writes are successful?	242
How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes?	243
How do I create an audit in the Event Log for user logins and logouts?	243
Why do I get an "out of memory" message when I try to run the Automation Interface?	243
Why am I seeing #COM for circuit breaker status in the genie status page?	244
Why can't I acquire waveforms in the waveform viewer?	244
Why won't the Excel DBF Add-In toolbar install?	244
What causes the "First dbf record" error message? How do I keep it from happening?	244
Why is my device in comms loss?	244
Index of Terms	245

Hazard Categories and Special Symbols

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



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



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

DANGER

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

WARNING

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

CAUTION

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

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.

Safety Precautions

WARNING

HAZARD OF INCORRECT INFORMATION

- Do not incorrectly configure the software, as this can lead to incorrect 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, or equipment damage.

WARNING

LOSS OF CONTROL

- The designer of any control scheme must consider the potential failure modes of control paths and, for certain critical control functions, provide a means to achieve a safe state during and after a path failure. Examples of critical control functions are emergency stop and overtravel stop.
- Separate or redundant control paths must be provided for critical control paths.
- Do not ignore alerts during device driver installation. If you choose to ignore such alerts, the driver will be installed but may operate incorrectly.

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

Introduction

Welcome to the help file for PowerSCADA Expert, version 7.30. 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 11

[*Assumptions*](#) on page 11

[*What is SCADA?*](#) on page 11

[*What is PowerSCADA Expert?*](#) on page 11

[*Prepare for PowerSCADA Expert*](#) on page 11

[*Components of a Project*](#) on page 12

[*Configuration Tools*](#) on page 12

[*Runtime Environment*](#) on page 12

[*Supported Device Types and Protocols*](#) on page 12

[*System Requirements and Installation Instructions*](#) on page 15

[*Uninstall and Reinstall PowerSCADA Expert*](#) on page 15

[*Quick Start: Typical Workflow*](#) on page 16

[*IEC 61850 System Setup Workflow*](#) on page 19

[*Get More Information*](#) on page 20

[*Manuals*](#) on page 20

[*Help files*](#) on page 20

[*Customer support*](#) on page 20

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 7.30 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 11](#)

[What is SCADA? on page 11](#)

[What is PowerSCADA Expert? on page 11](#)

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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help). 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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help). The project components include graphic components, tags, alarms, system components, communications components, I/O Server components and Cicode/CitectVBA.

Configuration Tools

Configuration tools consist of:

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

Profile Wizard: Using this wizard, you will import device profile information from the Profile Editor into a 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
- MicroLogic 5P and 6P, A, H
- CM4000 series
- PM800 series
- PM700 series

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

How Do Drivers Work?

The following paragraphs describe driver subscriptions.

Two Subscription Types

Are there two different types, one used between the graphics level and I/O server, and one for polling devices and cache refreshing? No, there is only one subscription that begins at a client system and ends up in the I/O server. 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 etc. 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, does the cache still get refreshed? 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.

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 55](#) 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 57](#) 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 *PowerSCADA Expert 7.30 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 7.30 (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 7.30 (for Vista operating systems or later, it is [Project Drive]\ProgramData\Schneider Electric\PowerSCADA Expert 7.30. 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 26](#) 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 22](#).

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 93](#) 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 122](#).

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 103](#)) to add all of the devices from one page (example found in: Program Files\Schneider Electric\PowerSCADA Expert 7.30\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 120](#).

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 122](#) 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. **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.

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 60](#)).
2. Import the first ICD file into the Profile Editor (see [Import SCL Files on page 73](#)).
 - 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 IEC61850 Device to a Project on page 99](#)).
 - a. If needed, add/edit datasets and report control blocks (see [Edit IEC 61850 Datasets on page 55](#) and [Edit IEC 61850 Report Control Blocks on page 57](#)).
 - 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 IEC61850 Device to a Project on page 99](#).
10. Compile and run the project.

Get More Information

For additional information, use one of these options:

[Manuals on page 20](#)

[Help files on page 20](#)

[Customer support on page 20](#)

Manuals

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

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

Help files

In addition to the help file released with this product, there are several related help files. They are located in the PowerSCADA Expert Bin folder: \Schneider Electric\PowerSCADA Expert 7.30\Bin).

Customer support

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

Asia/Pacific: +852 2579 9568

Europe/Africa: +33 4 76 60 62 35

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

web support: www.powerlogic.com

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 the Citect installation medium under Extras > Migration Utility.
3. Run the file named *PSEMigrationUtility*:
 - a. 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
Populate Tag Prefix- Equip.dbf	Equipment Name, which was used to build tag names, is now the equipment hierarchy name (can non 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 @(Protection), 2, "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 @(Protection), 2 the "2" is removed; it will be changed to @ (Protection).

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 the Citect installation medium under Extras > Migration Utility.
3. Run the file named *PSEMigrationUtility*:
 - a. 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
Populate Tag Prefix- Equip.dbf	Equipment Name, which was used to build tag names, is now the equipment hierarchy name (can non 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 @(Protection), 2, "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 @(Protection), 2 the "2" is removed; it will be changed to @ (Protection).

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 25](#)

[Typical Workflow Illustration on page 26](#)

[Launch the Profile Editor on page 30](#)

[Add Engineering Unit Templates, Units, and Conversions on page 32](#)

[Define Device Types and Tags on page 36](#)

[Device Type Screens and Workflow on page 36](#)

[Print the .CSV File on page 41](#)

[Add Custom Tags and Tag Addresses on page 44](#)

[Add, Edit, or Delete a Device Type on page 38](#)

[Edit Tag Addresses on page 46](#)

[Edit Generic Tag Addresses on page 50](#)

[Edit Functional Addresses on page 43](#)

[Create Device Profiles on page 51](#)

[Enable Waveforms on page 51](#)

[View Device Profiles on page 51](#)

[Add Edit or Delete Device Profile on page 52](#)

[Edit IEC 61850 Datasets on page 55](#)

[Edit IEC 61850 Report Control Blocks on page 57](#)

[IEC 61850 System Setup Workflow on page 19](#)

[G3200 Device Setup on page 60](#)

[Set Up Projects in the Profile Editor on page 62](#)

[Project Screens and Workflow on page 62](#)

[The Set Up Projects Tab on page 63](#)

[Add, Edit, or Delete a Project on page 63](#)

[Create a Composite Device Type](#)

[Edit and Delete Information in a Project on page 65](#)

[Customize Tag Names on page 65](#)

[Import and Export Project Files on page 66](#)

[Export a Project on page 66](#)

[Profile Editor Export on page 68](#)

[SCL Export on page 68](#)

[Import Files into the Profile Editor on page 69](#)

[Using Import Templates on page 73](#)

[Tag Types on page 75](#)

[Customize Tag Names on page 65](#)

[IEC61850 Tag Construction on page 75](#)

[Define an Enumeration on page 76](#)

[Format Code Definitions on page 77](#)

[Logic Codes on page 79](#)

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 36](#) 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 51](#) for complete instructions.

Use the Set Up **Project** tab and its screens to bring together all of the system attributes for a single customer or installation. For example, the customer installation will include a certain combination of device profiles (depending on the devices installed at the site). The project allows a specific unit template to be applied, converting units (such as watts) into units used by the customer (such as megawatts). This causes tags to display in the converted format. Projects also allow you to rename tags to suit a customer's needs (for example, Current A could be renamed to Current Phase A). See [Set Up Projects in the Profile Editor on page 62](#) for complete instructions.

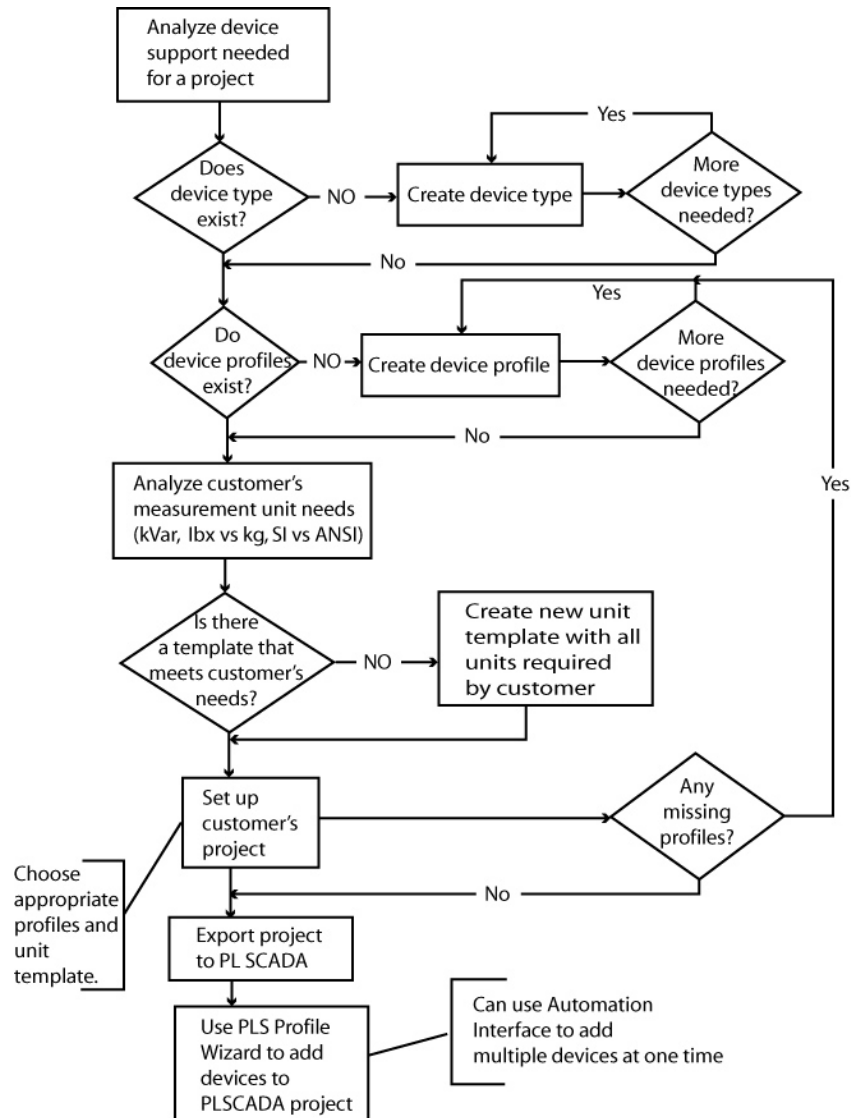
This product uses the IEC61850 tag-naming convention to create tags that measure device quantities. Although most of the tags you will use are already entered into the system, you can add custom tags. For more information, see [Tag Types on page 75](#).

Typical Workflow Illustration

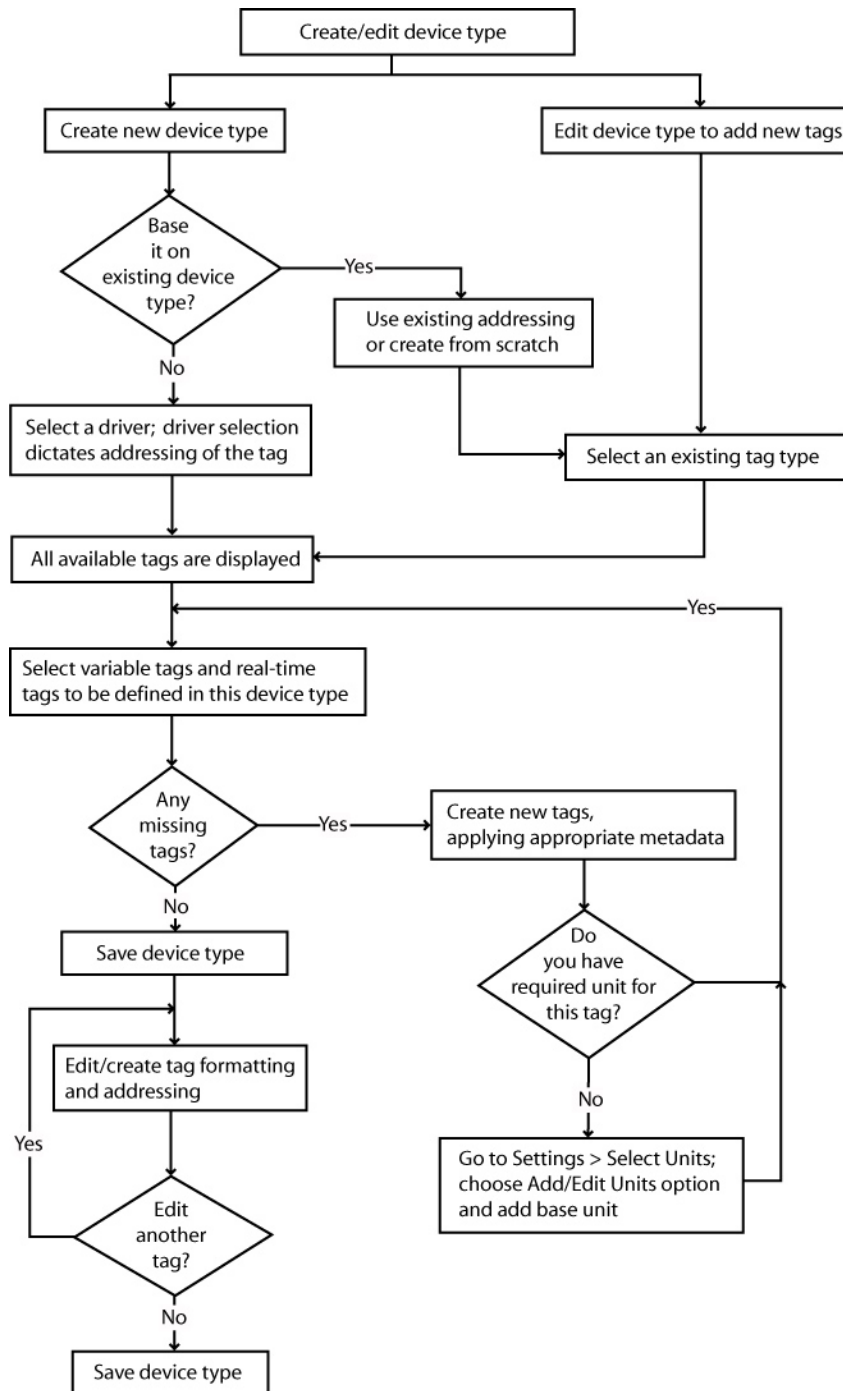
The following four flow charts illustrate the use of the Profile Editor. The first illustration provides an overview, and the following illustrations show:

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

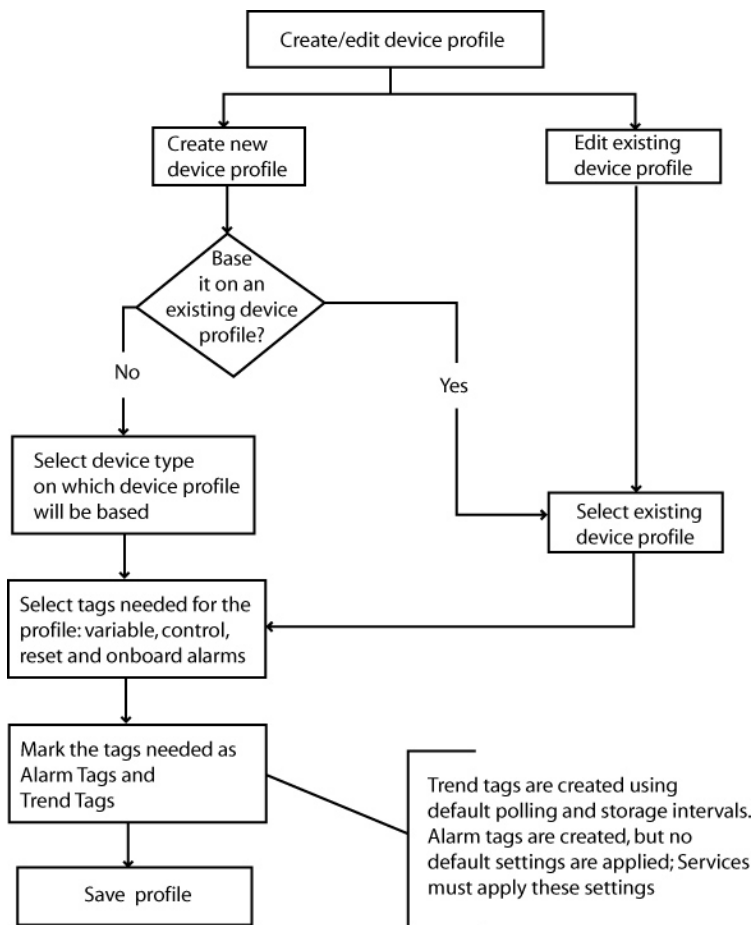
Workflow Overview



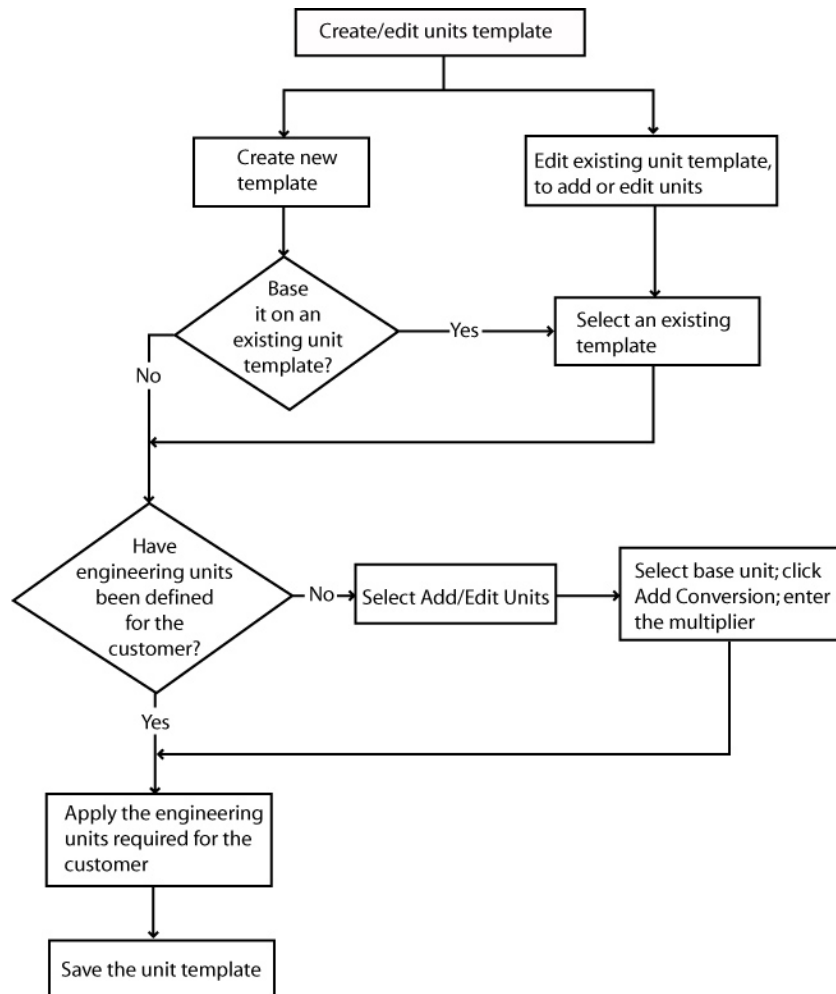
Create/Edit Device Type



Create/Edit Device Profile



Create/Edit Unit Templates



Launch the Profile Editor

To launch the Profile Editor, click Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > 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.

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 66 .
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 66 .
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 73 .
Settings > Set Up Custom Tags	Displays the Add/Edit Custom Tags screen. See Set Up Custom Tags on page 44 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 42 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 32 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 55 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 32](#)

[Add or Edit a Base Engineering Unit or Conversion on page 34](#)

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.

Field Name	Valid Entries	Comments
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 34 .
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 34 .
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 34 . Fahrenheit to Celsius temperature conversions must be handled by editing Cicode (Citect.ini).
Abbreviation	n/a	This is abbreviation for the selected unit. When the Selected Unit is changed, this field changes accordingly.
Multiplier	n/a	Added for the unit and for the conversion when the base unit was created. Pre-defined units/conversions cannot be changed. To edit a user-created unit, see Add or Edit a Base Engineering Unit or Conversion on page 34 .
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	- From the dropdown list, choose the unit you want to use for this tag/tag group. - Click the radio button for the exception to be made. - Either double-click the tag, or click the right arrow to move it to the Exception list.	- If you choose Apply Unit Conversion, the tag will be reported according 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. - 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 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 34](#).

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 34](#)).
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.

Field Name	Valid Entries	Comments
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, e.g., 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
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 36](#)

[Use the Define Device Type Tags Tab on page 36](#)

[Add, Edit, or Delete a Device Type on page 38](#)

[Print the .CSV File on page 41](#)

[Set Up Device Type Categories on page 42](#)

[Edit Functional Addresses on page 43](#)

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 34](#).
2. To manage the tags you will use, see [Define Device Types and Tags on page 36](#).
3. To add and edit custom tags, see [Add Custom Tags and Tag Addresses on page 44](#).
4. To add or edit device types, see [Add, Edit, or Delete a Device Type on page 38](#).
5. To establish device type categories and subcategories, used in reporting, see [Print the .CSV File on page 41](#).
6. To edit tag addresses, see [Edit Tag Addresses on page 46](#).

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 75](#).

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.

Field Name/Valid Entries	Comments
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 30 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.
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 75 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 75 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 46 . for detailed information.
Register 1/Display only (Real Time tags only)	This field contains first register used to store this tag. If there are additional registers, they are indicated in the address. The total number of registers is listed in the Num Registers column. This field allows you to verify and/or change the value of Register 1 without having to open the Edit Address screen. Note: If you enter a number that is not compatible with other address 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.

Field Name/Valid Entries	Comments
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 46 .
Scaling Register/View or enter the register number (Real Time tags only)	This is entered in the Edit Address screen, but it can be edited here. It is the register used to read the value for scaling. Note: If you enter a number that is not compatible with other address settings, you are prompted to go to the Edit Address screen.
Functional Address/Display only (Real Time, Onboard Alarm, Control, and Reset tags)	If you have added a functional address for this tag, it displays here. To add or edit this address, use the Edit Functional Address field. Note: Functional addressing is described in Appendix 4: Glossary on page 221 .
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" 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 44 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 44 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.
Device Category	Choose the category for this device.	To create categories, see Print the .CSV File on page 41 . In addition to predefined categories, you can add custom categories. See Set Up Device Type Categories on page 42 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 (100ms, Onboard Alarm, Power Factors, etc.).
Selected Tags	Select tags from Tags; click the right arrow to move them to this box.	You can move single tags or entire tag groups. They must be moved one at a time (cannot shift+click to select). Note: You cannot deselect tags for a device type if that device is associated with a device profile.

Field Name	Valid Entries	Comments
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 44 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 43](#)).
Real Time Tags sub-tab: you can edit the address (see [Edit Tag Addresses on page 46](#)) 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:

- From the Define Device Type Tags tab, click Add/Edit.
- 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.
	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

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

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

1. Tag Name and Original Address: These fields display from the tag you selected; you cannot edit this information.
2. Device Variables: Click New to begin adding new variable properties. The following fields become live:
 - a. Name This name must be in format %NNN%, where NNN includes only letters or underscores.
 - b. Description: This required field is free-form. It displays in the Profile Wizard and will help you ensure that you have the correct information entered.
 - c. Regular Expression: You can use one of the pre-defined expressions, or you can create your own
 - d. Test Value: This will become the default in Citect; use it here for testing the new address.
 - e. Help: Use this optional field to add more definition to this address. It displays in the Profile Wizard.
3. Code Body: Enter the code in C# to define the action you want to take place.
4. Return: Type the return statement that you want from C# code. It might look like:
`string.Format("SomeString{0}SomeOtherString", someVariable)`
5. Result: Click Test in the lower right corner of the screen. If there is a compile error, check your C# code. Otherwise, the result displays. Verify that it is what you wanted.

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 46](#)

[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 46 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 (100ms, controls, currents, etc.) plus onboard alarms, resets, and controls.

Field Name	Valid Entries	Comments
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 79 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. 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.

Field Name	Valid Entries	Comments
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 75](#) and [Logic Codes on page 79](#).

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 79](#). Also, be sure you use the correct case. For example, use **M** for register numbers in hexadecimal, and use **m** for register numbers in decimal.

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

Real-Time Tag Addresses

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

Field Name	Entry	Comments
Data Type	For display only	You can edit this field in the Add/Edit Custom Tags screen.
Priority	High, Normal, or Low Logic Code:	You can edit this field either here or in the Add/Edit Tag screen.
Logic Code	Select the logic code for this tag.	The logic code list depends on the Data Type for this tag. For more information about logic codes, see Logic Codes on page 79 .
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.

Field Name	Entry	Comments
Invert Result	Check this box to invert.	Will turn False to True or vice versa; typically used for Normally Open or Normally Closed.

Onboard Alarm Tag Addresses

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

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

Reset Tag Addresses

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

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

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

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

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

Box Name	Field Name	Comments
Tag Information	Command Type	The Command Type and Command to Edit are already selected.
	Command to Edit	
Data Information box	Data Type: for display only	You can edit this field in the Add/Edit Custom Tags screen.
	Priority: High (default)	Cannot be edited.
	Logic Code: Select the logic code for this tag.	Choose the appropriate logic code for this tag. See Logic Codes on page 79 .

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 79 .

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 (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. See [Variable Tag Properties](#) for detailed information.

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 36](#)). 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 51](#)

[View Device Profiles on page 51](#)

[Add Edit or Delete Device Profile on page 52](#)

[Select Trend Intervals on page 55](#)

[Set Up Trend Intervals on page 55](#)

[Edit IEC 61850 Datasets on page 55](#)

[Edit IEC 61850 Report Control Blocks on page 57](#)

[IEC 61850 System Setup Workflow on page 19](#)

[Create a Composite Device Type on page 58](#)

[G3200 Device Setup on page 60](#)

[IEC 60870-5-104 Protocol Support on page 61](#)

[DNP3 Protocol Support on page 61](#)

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 SFT 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 [Viewing 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 51](#) for more information). To begin making changes, click Add/Edit. The following table describes the fields on this tab. The tags listed assume that Advanced Properties has been checked. Not all elements appear on every sub-tab. Detailed instructions are after the table.

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

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

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

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

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 65](#).
10. When you have all of the devices you want, click Next.
11. From the Device Type Tags list on the left, select the tags you want to include in this profile. You can select entire tag groups or individual tags from a group; but you must select them one at a time.
12. After each addition, the tag or tag group displays in the Selected tags box. You can override any tag name (typically for generic I/O devices with multiple tags, such as inputs, for which names alone would not be intuitive in runtime. To override a tag, select it, then click Override Tag Name. Choose the tag you want. Click OK. The new tag will correctly display the value of the original tag, but will take the appearance of the override tag (description, metadata, etc.).
13. The final column, Is Device Tag, displays only for composite devices. Check this box to tie a tag back to its actual physical device. For example, if the same tag is in three devices, and you set PC-based alarms for each device, you need to be able to determine which device has a problem in runtime. To prevent confusion, check Is Device Tag to cause 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 55](#). To add additional trend tags, see [Set Up Trend Intervals on page 55](#).

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 55](#) and [Edit IEC 61850 Report Control Blocks on page 57](#) 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 55](#)).
 - 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 44](#)).
 - Onboard Alarms sub-tab: enable waveform capture for on-board alarms (see [Enable Waveforms for Onboard Alarms on page 141](#) 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 44](#) for more information).
 - Parameters sub-tab: Edit parameters for IEC 61850 driver parameters (see [Edit Driver Parameters on page 13](#) 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 55](#).

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. See [Trend Tag Properties](#) in the CitectScada.chm help file for details on each field.
5. Click OK to save the new trend definition, which displays in the left-hand box.

To edit a trend interval:

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

To delete a trend interval:

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

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.

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 Add/Edit Device Profile screen, give the composite device type a name. Optionally, add a description (which will become a tool tip display in later screens). Click Next.

4. Choose the device types to be in the composite. Click Next.

The Add/Edit Device Profile displays with only device type tags available for selection.

5. Add the tags you need for each device type listed on the left. To add all of the tags for a device type, highlight the device type name and click the right green arrow.

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

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

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

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

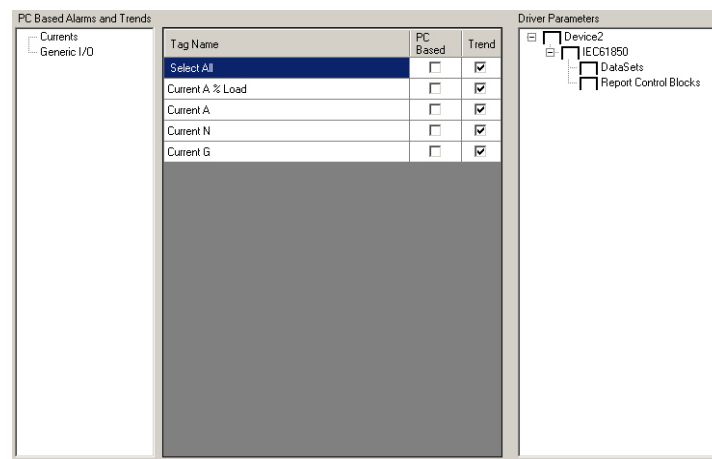
When the project is added to the project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the project.

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 for information on editing parameters.
12. Check the Close Wizard box, and click Finish to return to Create Device Profiles tab/ Or, leave it unchecked, and click Finish to return to the Add/Edit Device Profile screen.

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, etc.)
4. Create the project that will include the GS3200 (mark profiles under the G3200 as data concentrated devices).
 - a. Ensure that "Add As Default" is not checked for the project.
 - b. Add the first device profile.
 - c. At the Select Profiles screen, enter the Configured Name, and check the "Data Is Concentrated" box.
 - d. Continue with steps "b" and "c" for additional device profiles.
5. Run PowerSCADA Expert and SCL exports.

In SFT850

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 SFT850.
 - c. Complete the remaining steps in the Profile Wizard.
3. Add a new device for each device under the G3200.
 - a. From the Enter instance information screen, change the logical device as needed. Select the unit name of the G3200 device for the data concentrator.

For use with a single device

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

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

In the Profile Editor

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

In SFT850

Create the CID file for the G3200 gateway.

In 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 SFT850.
 - d. Complete the remaining steps in the Profile Wizard.

IEC 60870-5-104 Protocol Support

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

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

1. To obtain the license key, contact your local sales representative. Add the license to the dongle. If you need more information, see [CiUSAFE dialog properties](#) in the citectscada.chm help file.
2. The IEC 60870-5-104 driver is included in the install executable: Extras\Drivers\Specialty Drivers\IEC870-5-104.
3. Run the Profile Editor support utility. The .exe file is in Program Files > Schneider Electric > Profile Editor\v7.30\Add IEC 104 Driver Support.exe.

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

DNP3 Protocol Support

You can create device types and profiles that use the DNP3 protocol. See [Add a DNP3_TCP Device on page 97](#) or [Add a Serial Device to a Project on page 95](#) 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 62](#)

[The Set Up Projects Tab on page 63](#)

[Add, Edit, or Delete a Project on page 63](#)

[Create a Composite Device Type on page 58](#)

[Edit and Delete Information in a Project on page 65](#)

[Customize Tag Names on page 65](#)

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 the 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 65](#).
- 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 65](#).

To add or edit project information, click Add/Edit. The Add/Edit Project screen displays. See [Add, Edit, or Delete a Project on page 63](#) 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 66](#) 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 32](#) 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 34](#) 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 221](#).)



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, etc.) in future project references.
 11. When all profiles are added, click Save to save the changes, or click Save & Exit to save changes and close the screen.

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 89](#) 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 66](#).
- 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 68](#).
- 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 68](#).
- Import a project from another instance of the Profile Editor.
- Import SCL files. This allows you to import from the profile data of IEC61850-compliant devices and create device types. These files can be exported as an IID profile or as a PowerSCADA Expert profile.

The import process works the same for each type of import. The only exception is that you cannot import profiles when you are importing SCL files. See [Import Files into the Profile Editor on page 69](#).

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 73](#).

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 7.30 > 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\v7.30\Projects (for Vista operating systems or later, it is ProgramData\Schneider Electric\Profile Editor\v7.30\Projects).
- Each Profile Wizard profile file is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.30\WizardProfiles\[project name] (for Vista operating systems or later, it is ProgramData\Schneider Electric\Profile Editor\v7.30\WizardProfiles\[project name]). A single file for each included profile.
- The Equipment.Profiles file (contains all of the Profile Wizard profile information and the base profile information used by the Profile Wizard) is stored in Program Files\Schneider Electric\Profile Editor\v7.30\WizardProfiles\[project name]\Profile Wizard.

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

In 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 7.30\User\[Project] (for Vista operating systems or

later: [Drive Letter]:\ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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 144](#).

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 65](#) and [Add Project Parameters on page 65](#) for the information you need to make the changes that you want.
5. Click Export Project.

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

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

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

C:\Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\7.30\WizardProfiles\YourProjectName\
Profile Editor Export\YourProjectName.pls.

SCL Export

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

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

The only ways you can edit these files are:

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

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

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:

C:\Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.20\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 specific information about importing SCL files, see [Import SCL Files on page 73](#)

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 SCL files 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 73](#)

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

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

See [Import Reconciliation Screen \(Search\)](#) on page 70 for a description of the parts of this screen.

7. As you view the Import Reconciliation screen, you work through the items in the left-hand tree. You respond to each item according to your preference for it. You must match, merge, or reject first units, then tags, and finally the device type.
8. For each item in the left-hand pane, when you match it with a local item, the right-hand pane changes to allow you to edit how you want the match to take place.

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

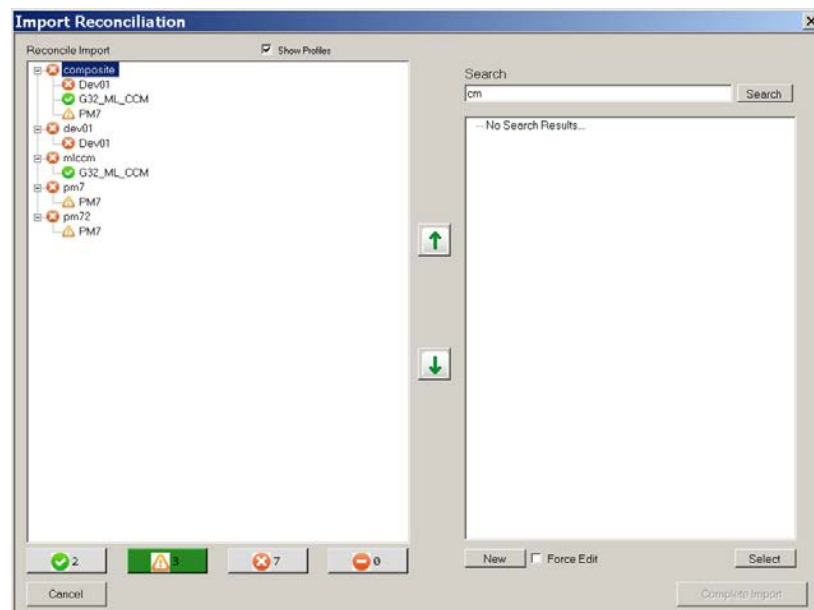
See [Import Reconciliation Screen \(Merge-Match\)](#) on page 71 for instructions on responding to each type of tag.

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

After the import is saved, the Save Import Template dialog displays. See [Using Import Templates](#) on page 73 for instructions on creating, using, and deleting import templates.

Import Reconciliation Screen (Search)

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



The parts of this screen are:

Left-hand pane:

- The tree view presents the imported file's hierarchy: device profiles, device types, tags, and units. Each item has an icon to its left, indicating the system's initial assumption about how this item is matched. You can accept or override these assumptions.
- The Show Profiles box controls whether the tree view lists device profiles (it is possible that device profiles will be discarded during the import).
- At the bottom of the page, the four buttons control how the tree navigates. (Each individual item also has one of these icons beside it, indicating its status.) You can select the items singly or in combination (click Ctrl +). The buttons are:
 -  : exact match; item is either a perfect match to a local item, or you accepted a merge for it.
 -  : partial match; item is a partial match to a local item.
 -  : no match; item does not match any local item.
 -  : ignore this item; you chose to ignore this item in this import.
- Cancel: Click to back out of the import without saving any of your selections.

Right-hand pane:

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

Edit the Item

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

Complete the Import of an Item

When you make a match—exact or partial—between an imported item and a local item, the right pane of the screen displays information about the match. See [Import Reconciliation Screen \(Merge-Match\) on page 71](#) for more information.

Import Reconciliation Screen (Merge-Match)

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

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

Partially Matched Device3

Item requires verification

Property	Imported	Local	Result
DeviceTypeDisp...	Device3	CM4-w-custom	CM4-w-custom
DeviceTypeName	Device3	CM4-w-custom	CM4-w-custom
Category	Schneider Elect...	Schneider Elect...	Schneider Elec
Subcategory	Monitoring Devi...	Monitoring Devi...	Monitoring Dev
CM4000		CM4000	Local
IEC61850	IEC61850		Imported

Unmatch Merge Match

Complete Import

The parts of this window are:

- Title: Indicates which of the three "states" the import item is in: matched (the item name displays), partially matched (you will need to merge the two items or accept the match that is in the final column), or merging (the system assumes a match in the final column, but you can edit each attribute). These actions to take for these three states are described below.
- The columns describe the imported vs local items:
 - Property: The property for which the other columns provide definitions
 - Imported: The value of the item in the import file
 - Local: The closest local match for the imported item
 - Result: The item as it will be added in this import
- Unmatch: Removes the match for the item; if you were merging, the merge grid goes back to its original state for you to either reject it, find a match, or import as new.
- Merge: When the import item is matched with an unlocked item, click here to begin merging their attributes. The "Merging" heading displays, and you can begin choosing the setting to be used for each attribute.

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

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

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.

See [Import Files into the Profile Editor on page 69](#) for instructions on importing.

Using Import Templates

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

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

Creating a New Template During Import

To create a new template:

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

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:

[IEC61850 Tag Construction on page 75](#)

[Define an Enumeration on page 76](#)

[Format Code Definitions on page 77](#)

[Logic Codes on page 79](#)

[Block Writes on page 86](#)

IEC61850 Tag Construction

IEC61850 tags are flexible, which allows them to specify how functions are implemented in devices. The IEC61850 tag was developed for medium-voltage and high-voltage applications, such as monitoring, control, and substation automation.

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

If you are writing Cicode (see [Use Cicode to Customize a Project on page 145](#)). You will need to know the IEC61850 tag name that you added to the device profile for that device. You can print the .csv file to view tag names (see [Print the .CSV File on page 41](#)). 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)

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

The following example illustrates the IEC61850 tag for current A:

EquipmentName\MMXU1\A\PhsA

where:

M = the category

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

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

A = the data object, current

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

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

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
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 46](#) for instructions on editing the address,
4. When adding a genie for the circuit breaker on the graphics page, choose from the default library (see [Appendix 1: Default Genie Library on page 187](#)), or create a custom genie (see [Create New Genies on page 128](#)).

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 79](#) 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 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

Attributes	Modifiers	Comments
		- 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 79

Alarm Format Code Definitions

Attributes	Modifiers	Comments
T (type) Required	ALM = alarm	
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 79. Write Address format: T:SS;m:##;L:P:101 Example: C:NO;T:SS;m:1234;L:P101;T:SS;m:3456;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	Pow-erSCADA 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)

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
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)
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) 2-register Argos	UT	LONG	2 sequential registers	No	Argos	The number of seconds since 01/01/2000 (00:00:00) register 1 = MSB register 2 = LSB
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 Each register has a range of 0 to 9,999

Logic Code	IEC Type	Pow-erSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						Result is: – $R4 \cdot 10,000^3 + R3 \cdot 10,000^2 + R2 \cdot 10,000 + R1$
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 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 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. 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	SS	DIGITAL	2 registers	No	CM/PM	Same as PL Digital Output SS except:

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
(L:P:25)						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. 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 bit anded 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 an analog input register. Second register is treated as a signed value. Third register can contain a value from –3 to 3 and will be used to scale the second register (R2*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.
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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
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) 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. 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. 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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
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: $\text{Avg}(R1 \dots Rn) * 10^{\text{scale}}$ 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 (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Sum with Scale (L:P:45)	MV/C-M	REAL	2 registers	Either (required)	CM/PM	Result is: $(R1 * 10^{\text{scale}}) + R2$ NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Sum with Scale 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 (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Vector Math (L:P:47)	MV/C-M	REAL	2 registers	Either (required)	Generic	Result is: $\text{sqrt}(R1^2 + R2^2) * \text{scale}$ 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.
Vector Math IEEE (L:P:48)	MV/C-M	REAL	4 registers	Either (required)	Generic	Result is: $\text{sqrt}([R1 R2]^2 + [R3 R4]^2) * \text{scale}$ where [] indicates IEEE32 representation 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	MV/C-M	REAL	1 register	No	CM4	Returns the IEEE power factor.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
(L:P:50)						
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 (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
SP40 Power Factor IEEE (L:P:52)	MV/C-M	REAL	1 register	No	SEPAM 40	Returns the IEEE power factor (converted from IEC mode). NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
ML Power Factor IEEE (L:P:53)	MV/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 (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Generic Power Factor (L:P:54)	MV/C-M	REAL	2 registers	No	Generic	$R2/\sqrt{R2^2 + R1^2}$ where: R2 = real power R1 = reactive power NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Generic Power Factor - IEEE32 variation (L:P:55)	MV/C-M	READ	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
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 (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
WRITES (these are write-only; see below for Read/Write codes)						
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.

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
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.
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 103](#) 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 89](#)

[Add the Cluster, Network Address, and Servers on page 89](#)

[When You Set Up More than two I/O Servers per Cluster on page 91](#)

[Using the Wizard to Add and Remove Devices on page 93](#)

[Add a Device to a Project on page 93](#)

[Add a TCP Device to a Project on page 93](#)

[Add a Serial Device to a Project on page 95](#)

[Add a DNP3_TCP Device on page 97](#)

[Add an IEC61850 Device to a Project on page 99](#)

[Port Names on page 100](#)

[Remove a Device From the Project on page 102](#)

[Edit Devices in a PowerSCADA Expert Project on page 101](#)

[Add Multiple Devices through the Automation Interface on page 103](#)

[Use the OLE Automation Interface to Add and Remove Devices on page 105](#)

[Examples on page 105](#)

[Command Line Interface on page 106](#)

[Interface Workflow on page 106](#)

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 7.30 > PowerSCADA Expert 7.30 Explorer. Three windows display:

- Citect Project Editor
- Citect Explorer
- Citect Graphics Builder

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

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

- at least one **cluster**: A cluster is a group of alarms servers, trends servers, reports servers, and I/O servers. For additional information about clusters, see [Clustering](#) in the CitectSCADA.chm help file (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help).
- a **network server address for each cluster**: The name added here is linked to the IP address. This is the address to be used by all of the servers in the cluster. When using the host computer as the server, you can use the IP loopback address: 127.0.0.1.
- at least one of each relevant type of **server**: There are four types of server: alarm, report, trend, and IO.

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

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

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

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

An **I/O server** is a dedicated communications server that exchanges data between I/O devices and 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 116](#).

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 92](#) 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 92](#) 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 89](#)). 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 7.30\User\ [your project]\(for Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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 89](#).

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

[Add a Device to a Project on page 93](#)

[Add a TCP Device to a Project on page 93](#)

[Add a Serial Device to a Project on page 95](#)

[Add a DNP3_TCP Device on page 97](#)

[Add an IEC61850 Device to a Project on page 99](#)

[Port Names on page 100](#)

[Remove a Device From the Project on page 102](#)

[Edit Devices in a PowerSCADA Expert Project on page 101](#)

[Add Multiple Devices through the Automation Interface on page 103](#)

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 103](#) for instructions.

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

[Add a TCP Device to a Project on page 93](#)

[Add a Serial Device to a Project on page 95](#)

[Add a DNP3_TCP Device on page 97](#)

[Add an IEC61850 Device to a Project on page 99](#)

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 89](#)).

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*).

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.
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 7.30\User\[your project](for Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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 89](#)).

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*).
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.
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 7.30\User\[your project]

(for Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\User\[your project]). English.dbf contains terms that will be translated from English.

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

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

Click Next.

5. At the Select I/O servers screen, choose the primary and standby servers. You can add information for the standby server if you check the "Supports Redundancy" box. Click Next.
6. At the Configure Sub-Profile Communications Method screen, choose the communications method used for the first sub-profile in this project. Click Next.
7. At the Communications Settings screen, type all of the information for each server (com port, baud rate, etc.). If you check the "Same as Primary" box for standby, you will use the same addresses for the primary and standby. Click Next.
8. At the Port Settings screen, you can rename each of the ports. (See more information about naming ports in [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 89](#)).

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*).
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 7.30\User\[your project](for Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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 IEC61850 Device to a Project

Use this procedure when you want to add an IEC61850 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 89](#)).

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 91](#)).
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 7.30\User\[your project](for Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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 93](#) 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. 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 144](#) 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 89](#)).
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 7.30\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 7.30\Config.
Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\Config.
 - b. Cell B3: Type the project name.
 - c. Cell B4: Enter the path to the location of *equipment.profiles*:
For Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 7.30\User\ProjectName).
For Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\User\ProjectName).
6. From the Input worksheet at the bottom of the spreadsheet, enter the following information for each device that you want to add:
 - a. Profile Name: Type the complete name (to view exact addresses, open *equipment.profiles* in Notepad), e.g.:
SchneiderElectric.Protection Device.Sepam S42 Full

- b. Name: Enter the device name (no spaces or punctuation, limit of 32 characters). This becomes the equipment name in the PPowerSCADA 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	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

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 105](#)

[Command Line Interface on page 106](#)

[Interface Workflow on page 106](#)

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 7.30\Config\citect.ini";

myAuto.ReadProfiles(
@"C:\Documents and Settings\All Users\Application Data\Schneider
Electric\PowerSCADA Expert 7.30\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.

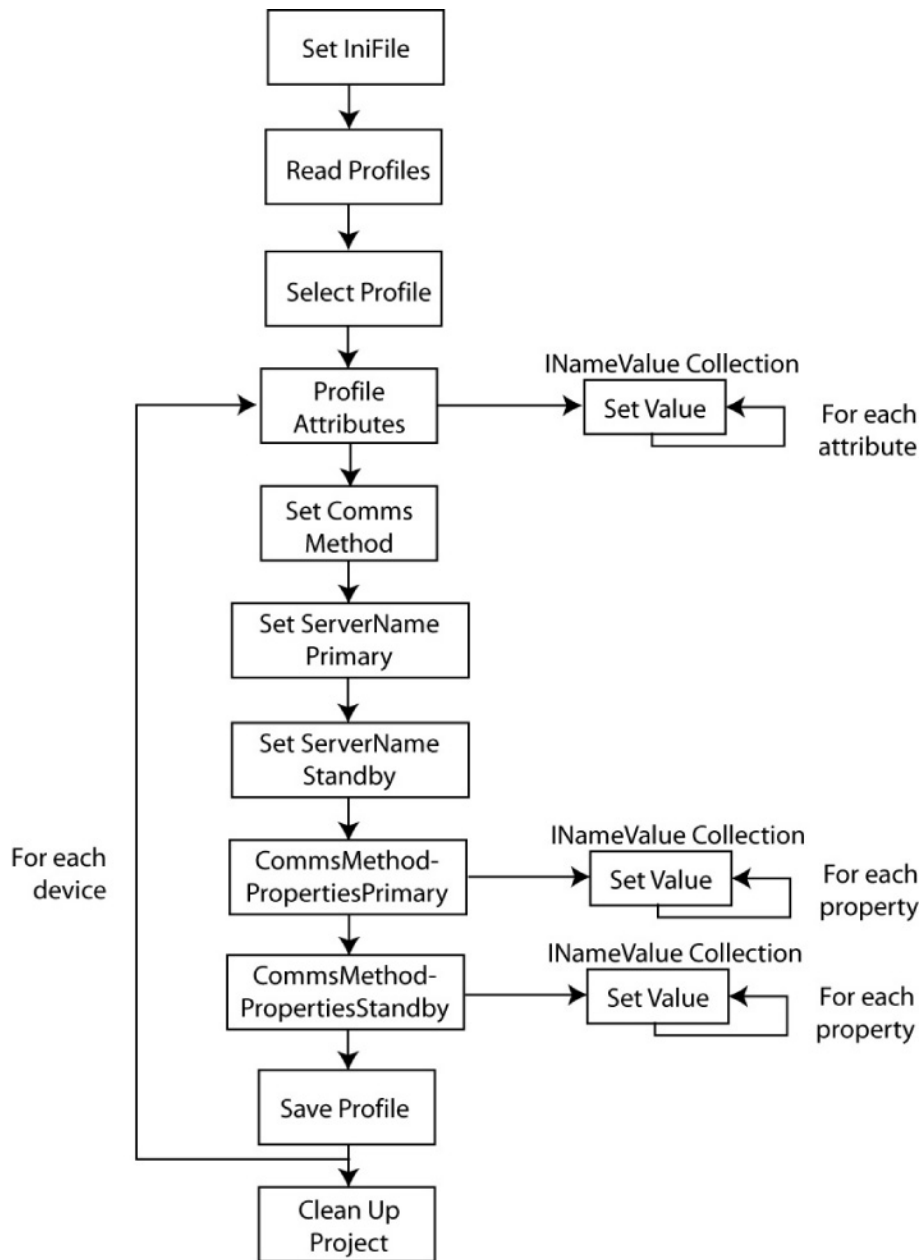
```
EquipmentWizard.exe /?
```

```
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.

Comms-Method	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.

Graphics: 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 112*](#)

[*The PowerSCADA Expert Pages on page 113*](#)

[*Create a Project on page 114*](#)

[*Reuse Projects Created in the Profile Editor on page 114*](#)

[*Before You Begin on page 114*](#)

[*Add a New Graphics Page on page 115*](#)

[*Create a One-Line on a Graphics Page on page 116*](#)

[*Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\) on page 117*](#)

[*Delete Information from PowerSCADA Expert on page 119*](#)

[*Use PowerSCADA Expert One-Lines*](#)

[*One-Line Flowchart on page 120*](#)

[*What Are PowerSCADA Expert One-Lines? on page 121*](#)

[*One-Line Device \(zOL\) on page 122*](#)

[*One-Line Colours on page 122*](#)

[*Add INI Settings to AdvOneLine.ini and Citect.ini on page 123*](#)

[*Start and Stop One-Lines on page 124*](#)

[*Genies*](#)

[*Genie Type Descriptions on page 125*](#)

[*Configure an Automatic Transfer Switch \(ATS\) on page 125*](#)

[*Configure a Busbar on page 125*](#)

[*Configure a Circuit Breaker or Switch on page 126*](#)

[*Configure a Meter on page 126*](#)

[*Configure a Source on page 127*](#)

[*Configure a Transformer on page 127*](#)

[*Create New Genies on page 128*](#)

[*GenieConfiguration.xml File on page 129*](#)

[*SupportedGenies.xml File on page 129*](#)

[*Configuration Utility*](#)

[*One-Line Configuration Utility on page 130*](#)

[*Review Genie Configurations on page 130*](#)

[*One-Line Errors and Warnings on page 131*](#)

[*Perform Repairs for One-Lines on page 132*](#)

[*Automatic Transfer Switch \(ATS\) Information on page 134*](#)

[*Busbar Information on page 135*](#)

[*Breaker and Switch Information on page 135*](#)

[*Meter Information on page 136*](#)

[*Source Information on page 137*](#)

[*Transformer Information on page 138*](#)

Work with Alarms on page 140

- Alarms Overview on page 140*
- Set Up Alarm Pages on page 140*
- Add Setpoints and Delays on page 140*
- Set Up an Alarm Based on an Enumeration on page 141*
- Change an Alarm Severity on page 141*
- Enable Waveforms for Onboard Alarms on page 141*
- Set Parameters for Event Log Length and Historical Logging of Events on page 142*
- Add an Onboard Alarm Tag on page 143*
- Set Up Audible Alarms on page 143*

Restore and Back Up Projects on page 144

- Restore a Project on page 144*
- PowerSCADA Expert Backup on page 144*

Use Cicode to Customize a Project on page 145

Time Zone Settings on page 146

Time Synchronization on page 146

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 117 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 (v7.30), 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 PowerLogic library of genies are provided. The genies contain logic such as control outputs and rack-in/rack-out. Once assembled and opened in the runtime environment, they provide an interactive view of an installation's power system.

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 117](#).

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 114](#)

[Add a New Graphics Page on page 115](#)

[Set a New Page as the Project Startup Page on page 116](#)

[Create a One-Line on a Graphics Page on page 116](#)

[Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\) on page 117](#)

[Create New Genies on page 128](#)

[Delete Information from PowerSCADA Expert on page 119](#)

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:
Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 7.30\User\[Project]
Vista operating systems or later: [Drive Letter]:\ProgramData\Schneider Electric\PowerSCADA Expert 7.30\User\[Project]
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 114](#).

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 63](#) 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 117](#) 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 Colour of Pages

You have the option of changing the background colour 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 colour backgrounds.
2. To designate background colour for each page, pick the colour in the Graphics Editor.

You can also change the genie font colours on your project pages. To do this see [Change the Genie Colour in Project Pages on page 115](#).

Change the Genie Colour in Project Pages

You have the option of changing the colour 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 colour you want to use. Make sure that there is sufficient contrast between the two colours so that the genie colours 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 colour of your project pages, see [Change the Background Colour of Pages on page 115](#).

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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help).

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

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



2. From the Paste Genie screen, choose a library (to ensure that all 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

Configure a Circuit Breaker or Switch

Configure a Meter

Configure a Source

Configure a Transformer

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 117 for help.

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 Configuration [KC_6-28]

Order: 1

Level 1: 2

Level 2:

Level 3:

Level 4:

Menu Command: 3

Symbol: 4

Comment: 5

Buttons: Add, Replace, Delete, Help

Status: Record : End of file

Menu Item	Description
1: Order	The relative position within the final graphics page. If you leave this field blank, the default value 0 is used. (64 characters max.)
2: Levels 1 through 4	These items establish the menu levels that will display. For example, you might use "Single Lines" for level 1, followed by the substation for level 2, and the graphic name for level 3. (Each line: 256 characters max.)
3: Menu Command	The Cicode expression that you want to execute. Typically, you will use the "page display" command followed by the actual page you want to see. For example: 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.

Delete Information from PowerSCADA Expert

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

Use PowerSCADA Expert One-Lines

Delete this text and replace it with your own content.

[One-Line Flowchart on page 120](#)

[What Are PowerSCADA Expert One-Lines? on page 121](#)

[One-Line Device \(zOL\) on page 122](#)

[One-Line Colours on page 122](#)

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 123](#)

[Start and Stop One-Lines on page 124](#)

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 130](#)

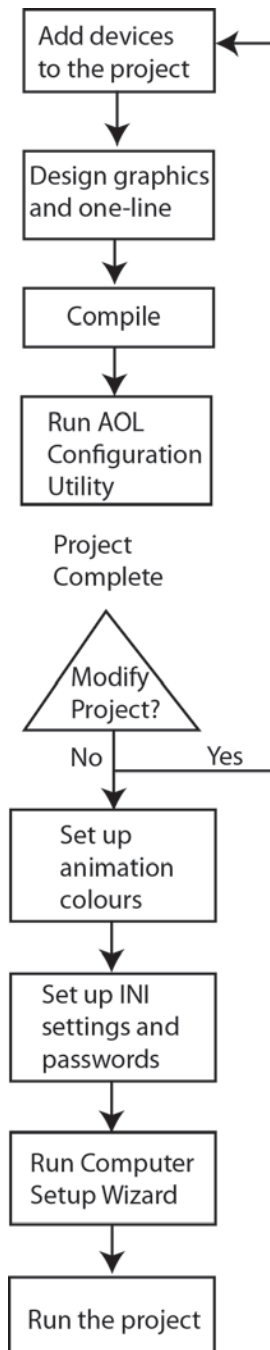
[Perform Repairs for One-Lines on page 132](#)

Setting Up Animation Colours:

[One-Line Colours on page 122](#)

INI Settings and Passwords:

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 123](#)



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

Ensure that all steps in the Quick Start procedure have been followed: [Quick Start: Typical Workflow on page 16](#).

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 colour "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. Select the device called *OneLine Device Setup*.
3. 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 tab.
3. Enter the optional parameters (MaxBreakers, MaxBuses, MaxMeters, MaxSources). Valid entries are from 1 to 9999 (only 200 for MaxSources).
4. Export the project.

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

One-Line Colours

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

In PowerSCADA Expert 7.30, colouring is based on the source and meter line active states. Sources dictate the colours for each genie. Meters can only determine if a bus is active. When the bus is live, the meter then colours based on the source that is connected to the bus. If there is no source, the default colour is used.

NOTE: Depending on how you configure transformers, you can either use this "pass-through" colouring, or you can use "voltage-level" colouring. See *Configure a Transformer* for more information.

Assigning Colours

To assign a colour 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 colours.
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 colour: click the colour cell for that source/transformer; then choose the colour from the dropdown list. You can also select a colour for unknown sources, off, and error. To indicate a flashing colour, you can select two colours.
4. When call colours are assigned, click Save.

Add INI Settings to AdvOneLine.ini and Citect.ini

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

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 possibly.	1000 msec
PrimaryServerIP	Used by redundant configurations to specify the IP address of the server on which the primary IOserver 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 OP address of the server on which the standby IOserver 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 nonfunctioning, 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 colour 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 colour that indicates that they are not being monitored. If an invalid colour 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.	6000 msec
StartupDelay	Sets the amount of time after AdvOneLine.exe has started for the system to be on line and all initializations complete.	15000 msec

Parameter	Description	Default Value
LoginUserName	This is the Citect user name to be used for the ctAPI connection in AdvOneLine.	N/A
LoginPassword	This is the Citect user password to be used for the ctAPI connection in AdvOneLine.	N/A
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 7.30\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

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
StartupCode	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.	[Updateinterval] + [TagSubscribeWait] * 5 milliseconds Minimum value: 1000 msec
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

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 122.

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 125

[Configure an Automatic Transfer Switch \(ATS\)](#) on page 125

[Configure a Busbar on page 125](#)
[Configure a Circuit Breaker or Switch on page 126](#)
[Configure a Meter on page 126](#)
[Configure a Source on page 127](#)
[Configure a Transformer on page 127](#)
[Create New Genies on page 128](#)
[GenieConfiguration.xml File on page 129](#)
[SupportedGenies.xml File on page 129](#)

Genie Type Descriptions

In the One Line Configuration Utility, each genie 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 135](#)
[Meter Information on page 136](#)
[Source Information on page 137](#)
[Transformer Information on page 138](#)

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 colours are determined, see [One-Line Colours on page 122](#).

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 colours are determined, see *One-Line Colours*.

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* 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*).
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 colours are determined, see *One-Line Colours*.

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 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* 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*).
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 colours are determined, see *One-Line Colours*.

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 colouring for the one-line. For more information on this, see *One-Line Colours*.
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 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 colours are determined, see *One-Line Colours*.

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 colouring for the secondary side busbar. Valid entries: 1–100.

There are two possible configurations:

Pass-through colouring: If you leave these fields blank, the transformer will transfer the colour 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 colour will be used).

Voltage-level colouring: When you enter top and bottom source numbers, the transformer colours 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 colour 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 colours are determined, see [One-Line Colours on page 122](#).

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 129](#).

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 129](#) and the [GenieConfiguration.xml File on page 129](#).

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 PowerLogic genies, including all of the smaller parts that you could use to create genies, see [Appendix 1: Default Genie Library on page 187](#).

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 colour during runtime (showing changing states). These genies are comprised of multiple symbols that have the same attributes (all call the same Cicode function, etc.).

1. Create the new symbol.
2. Configure the symbol to display the customized colours 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 colours. Please ensure that the default colour palette matches the palette used in PLS_Include. The parameter for *PLS_GetBusColorIndex(INT nColorIndex)* is the busbar number that determines the animated colour. Always use the %variable% notation. Using this notation permits you to reuse genies.

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

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

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 129](#)

SupportedGenies.xml File

[Example](#)

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

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 130](#)

[Review Genie Configurations on page 130](#)

[One-Line Errors and Warnings on page 131](#)

[Perform Repairs for One-Lines on page 132](#)

[Automatic Transfer Switch \(ATS\) Information on page 134](#)

[Busbar Information on page 135](#)

[Breaker and Switch Information on page 135](#)

[Meter Information on page 136](#)

[Source Information on page 137](#)

[Transformer Information on page 138](#)

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 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 Colours on page 122](#) 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 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 7.30. 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 7.30 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 135](#)

[Busbar Information on page 135](#)

[Meter Information on page 136](#)

[Source Information on page 137](#)

[Transformer Information on page 138](#)

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 colour. 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" colour (the array position 255 in the graphic).

Error Logging

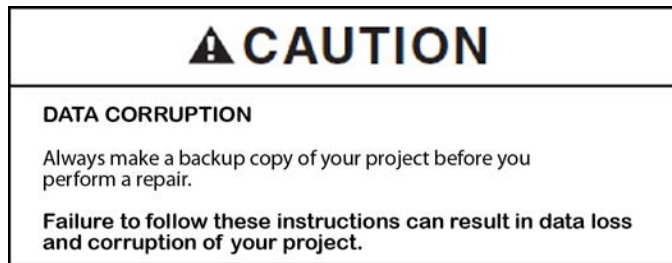
The most common errors in a CSV file are logged to the Run project in a file named AdvOneLineStatusLog.txt. The file can contain several messages. The table below lists these errors and their descriptions.

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: 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.



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 7.30, 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 7.30).

Repair option with Upgrade Project checked is used to upgrade projects from previous versions of the product. This option rennumbers all genies in the project. Do not perform this option on a project more than once, and do not perform it on PowerSCADA Expert 7.30 projects.

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

The following table describe 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 (✖)	

State	Solution
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).

Column	Description
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 (✖)	
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.

State	Solution
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 140](#)

[Set Up Alarm Pages](#)

[Add Setpoints and Delays on page 140](#)

[Set Up an Alarm Based on an Enumeration on page 141](#)

[Change an Alarm Severity on page 141](#)

[Enable Waveforms for Onboard Alarms on page 141](#)

[Set Parameters for Event Log Length and Historical Logging of Events on page 142](#)

[Add an Onboard Alarm Tag on page 143](#)

[Set Up Audible Alarms on page 143](#)

Alarms Overview

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

PC-Based Alarms

PC-based alarm tags are added in the Profile Editor, when adding each device profile. See for instructions. For instructions on entering setpoints and delays, see [Add Setpoints and Delays on page 140](#).

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 143](#).

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

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

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

- From the Analog Alarms window (accessible from the Project Explorer or Project Editor screens), you can type the setpoint and delay values for each alarm.
- When in the runtime environment, you can edit setpoints/delays that were set by the method above. See [View Waveforms on page 162](#) 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 76](#).

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 161](#) 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 be stored is controlled by the Alarm Summary length parameter, which defines the maximum number of alarm summary entries (Event Log entries) that can be held in memory. You can view these alarm summary entries on the Alarm Log page. Each event requires 256 bytes of memory, plus the length of the comment. 32,000 entries will require at least 8 MB of memory. If you have many events, you should ensure that there is enough memory to store them in RAM.

The default value is 5000.

NOTE: When the value is set to a number greater than 1000 for a multiple-cluster system, the alarm log might not display correctly. The list of alarm history that displays on a client might be shorter than the actual history stored on the alarm server. To avoid this problem, do one or more of the following:

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

Event Logging: [PLSEventLog] Parameter

Use this parameter to determine whether you will log Event Log entries and the file format that will 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.

NOTE: You can only add onboard alarms for devices using the CM4, PM8, Micrologic, or Sepam drivers. CM4, PM8, and Micrologic unique IDs must be decimal; SEPAM unique IDs must be hexadecimal.

1. From the device, obtain the unique identifier for this alarm. Additionally, for MicroLogic, you need to include the unique sub-identifier.
2. You also need the file number in which alarms are stored on the device.
3. From the Profile Editor, add the onboard alarm (see [Onboard Alarm Tag Addresses on page 48](#) 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 144](#)

[PowerSCADA Expert Backup on page 144](#)

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 68](#).

1. From the Citect Explorer, click Tools > Backup.
2. At the Backup Project window, select the project you want to back up
3. Browse to the location where you want to store the backup file.
4. In the Options box, check "Save configuration files." This saves the citect.ini file.
5. Click OK.

The backup .CTZ file is written to the following location: This is a Citect Zip file; you can open it with WinZip.

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 195](#).

For information about other parameters, see the [CicodeReference help file](#), located in Program Files\Schneider Electric\PowerSCADA Expert 7.30\Bin.

Customize Default Behaviours

In this section, you will find these topics:

[Time Zone Settings on page 146](#)

[Time Synchronization on page 146](#)

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

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

To learn about the parameters that you can use to configure drivers, see [Appendix 2: Citect INI Parameters on page 195](#).

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]\Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v7.30
Vista operating systems or later: [Project Drive]\ProgramData\Schneider Electric\PowerLogic\Profile Editor\v7.30)
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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help).

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 151](#)

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.

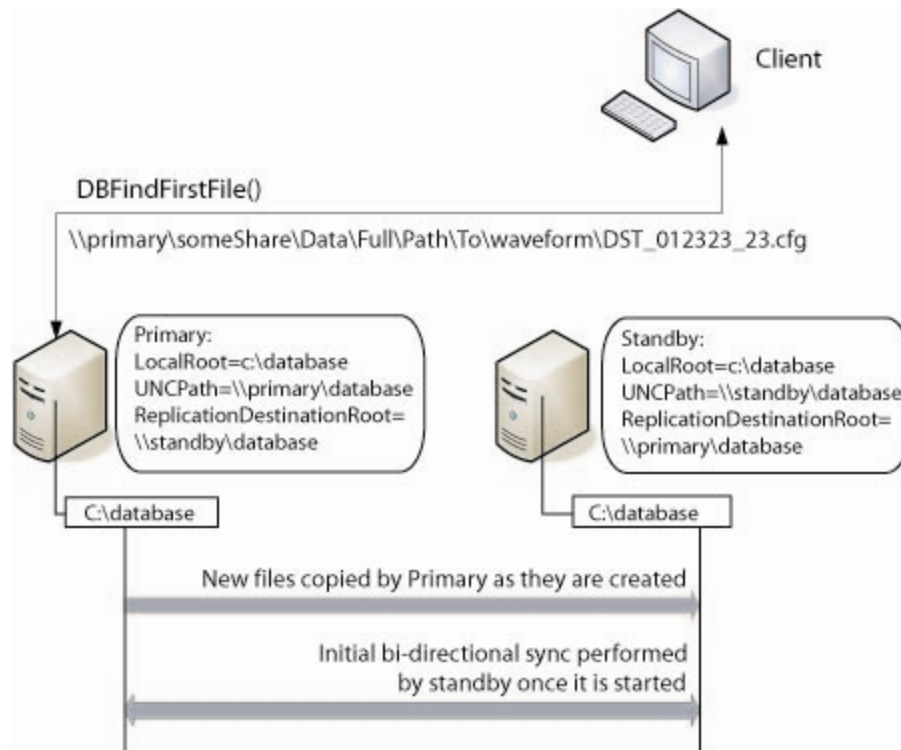
Waveform Database and Special Waveform Tags

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.



Graphics: 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 154*](#)
 - [*Log On to the Runtime Environment on page 154*](#)
 - [*View the Interface on page 154*](#)
 - [*Viewing the Alarms/Events Page on page 155*](#)
 - [*When Alarms do not Display Correctly on page 157*](#)
 - [*Event/Alarm Log Columns Table on page 158*](#)
 - [*Alarm/Event Filter Form on page 159*](#)
 - [*Use the Analysis Page on page 160*](#)
 - [*Use the Equipment Pop-Up Page on page 161*](#)
- [*PowerSCADA Expert Reports on page 174*](#)
 - [*Use Reports on page 166*](#)
 - [*Set up the Runtime Environment for Reporting on page 166*](#)
 - [*Set Up a Display Client for Report Viewing on page 167*](#)
 - [*Create and View Reports on page 168*](#)
 - [*Configure Email Settings to Send Reports on page 170*](#)
 - [*Send PowerSCADA Expert Reports via Email on page 171*](#)
 - [*Read, Export, Print, and Edit Reports on page 173*](#)
 - [*Single Device Usage Reports on page 175*](#)
 - [*Multi Device Usage Reports on page 175*](#)
 - [*Tabular Reports on page 176*](#)
 - [*Trend Reports on page 177*](#)
 - [*Enable Windows Authentication for Reporting on page 177*](#)
 - [*URL Routing for Reports on page 178*](#)
 - [*Report Localization on page 179*](#)
- [*Multi-Monitor Support on page 180*](#)
- [*The Demo Mode on page 181*](#)

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 7.30 >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 117.

In this section, you will find these topics:

[Log On to the Runtime Environment](#) on page 154

[View the Interface](#) on page 154

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

[Use the Analysis Page](#) on page 160

[Use the Equipment Pop-Up Page](#) on page 161

Log On to the Runtime Environment

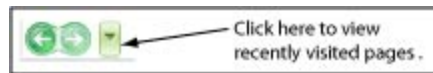
In the upper right-hand corner, click the Login link. Then enter your User ID and password. The features that are available will vary, depending on your user level.

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 117. 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 128 for details on customizing genies.

Communications Loss

When there is a communication loss for a device, the genie or any part of the genie on the one-line page should have cross-hatches (gray dots) over the affected area, and a communication loss (CL) indication displays on the genie. An alarm should also annunciate. The color state before communication was lost will remain unchanged.

However, the indication of loss of communications does not filter through the entire bus animation: the downstream part of the drawing may still appear as if communication is working. When any part of a one-line drawing loses communication, do not continue to trust downstream readings until you address the loss of communication.

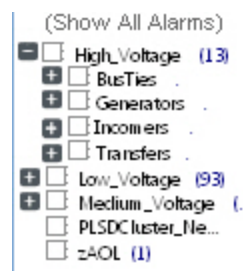
Viewing the Alarms/Events Page

See also: [Use the Equipment Pop-Up Page on page 161](#)

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.

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 159](#) 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 159](#)). From the Advanced Alarm Filter window, you can filter by date range, by text matches for various attributes, or by alarm type.

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 51](#)).

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.

Waveforms must be correctly set up before they will display. See [Enable Waveforms for Onboard Alarms on page 141](#) for more information. See [View Waveforms on page 162](#) 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. 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 7.30\Data Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\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.	

Issue	Cause	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.
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 159 .
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	only listed in event log, but I can make it show up in alarms
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

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:	
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.

Filter Option	Description: Display all alarms for:
	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>

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

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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help).

- Trending: Use this option to view historical trends. To select tags, click the Add Pen button on the toolbar:



Then associate the pen with a tag. By default, most trend data is polled every 15 minutes, and it is stored for one year (Citect Project Editor > Tags > Trend Tags) or until it is FIFO'd out. Some tags are polled every 5 seconds and are stored for two weeks. These tags are:

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

- Instant Trend: Use this option to view real-time trends. This allows viewing of data that is not set up for storage. To select tags for this trend, click the Instant Trend Selector on the toolbar:



NOTE: If one of the pens returns a value of "1.#R," the tag selected was not valid; no number could be reported for it. None of the values for any of the pens in the trend will be updated. To solve this issue, close the trend and open it again. This time, do not include the pen that gave the invalid return.

In either trending option, click Help for help using the tool:

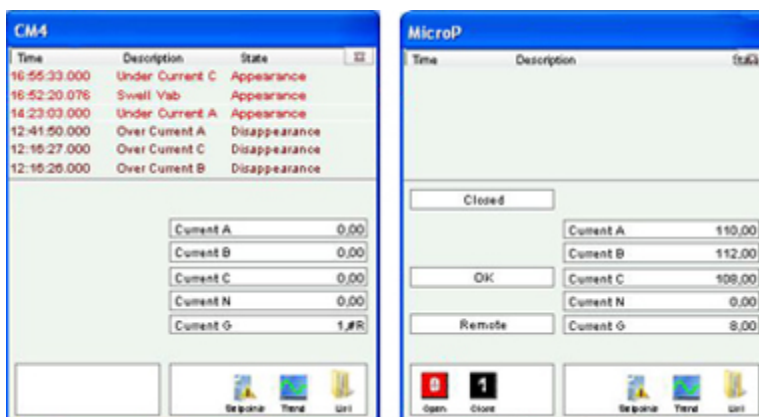


Use the Equipment Pop-Up Page

See also: [Viewing 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 184](#) for user access levels).

One of two status pages displays. The page on the left illustrates the status page for a meter genie. The page on the right illustrates the status page for a circuit breaker genie.



At the top of the page, the most recent alarms and events are listed (racked in/out, Comms Loss, etc.). To view details about an individual alarm or event, right-click the alarm. You can view:

- a waveform. (If you do not see "Waveform" in the list when you right-click the alarm, there are no waveforms for this alarm.) Waveforms can display only if the device is

set to “acquire on event,” and the waveform option is checked in the Profile Editor (see [Enable Waveforms on page 51](#)).

When the waveform is available for viewing, it displays when you click this link. Waveforms must be correctly set up before they will display. If there are multiple waveforms, you must select from the list that displays (by default, the waveform search returns all waveform files acquired within the 24 hours prior to the time of the alarm). See [Enable Waveforms for Onboard Alarms on page 141](#) for more information. See [View Waveforms on page 162](#) 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.

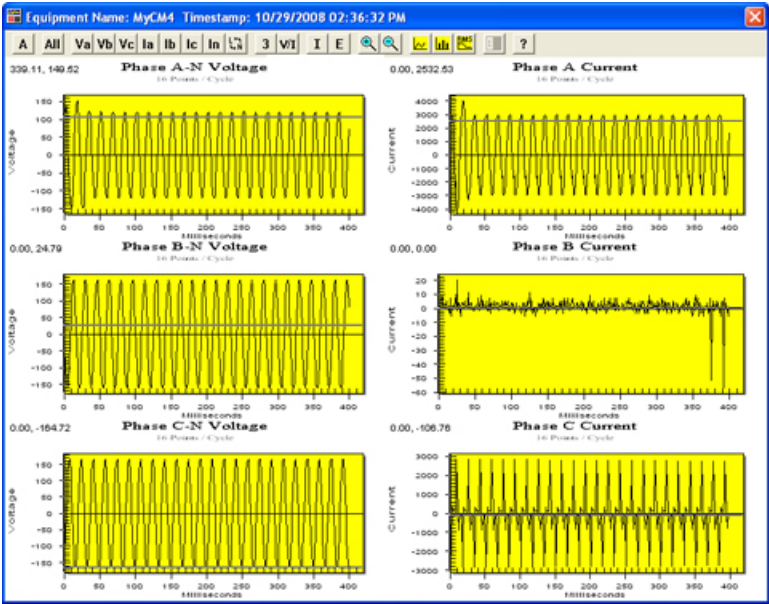
On the right side of the page, real-time values will display for the tag type that you chose in the Value field when you added the genie in the design-time mode. For example, if you enter MMXU1\A\phsA as the value, you will see real-time currents here, as illustrated above. If you did not enter anything in the Value field when adding the genie, this area will be blank.

At the bottom left corner of the circuit breaker status page, Open, Close, Reset (for circuit breakers).

At the bottom right corner, are the Setpoints, Trend, and List options. See the following sections for descriptions.

View Waveforms

After you select a waveform for viewing from the genie status page (and after the waveform has been sent to PowerSCADA Expert), the waveform displays, as shown below.



The toolbar at the top of the screen includes these buttons:

Button	Description
	Not currently used.
	Click to display all graphs (all three voltages and all three currents of the waveform capture).
	Click to display individual voltage phases (Va = A-N, Vb = B-N, Vc = C-N).
	Click to display individual current phases (Ia = phase A, Ib = phase B, Ic = phase C, In = neutral, Ig = calculated ground current).
	Click to toggle between line-to-line and line-to-neutral voltage phases.
	Click to view all three voltage or all three current phases within the same graph. Phase A, B, and C voltages display together. Phase A defaults to black, B to blue, C to red. In the same way, phase A, B, and C currents display together (A is black, B is blue, and C is red). To change colours, right-click the graph title. Select Control Properties. Go to the Colours tab and make changes.
	Click to view each phase, both voltage and current, within a single graph. You see phase A voltage and current in one graph, phase B voltage and current in another, and phase C voltage and current in a third.
	Click to display the Import Waveform dialog. If you have SMS installed on the computer, it defaults to the Comtrade folder. If not, it goes to the root of the C drive, for you to locate the waveform you want to import. Double-click the waveform to view it.
	Click to begin exporting a waveform. Select the type of file you will save (bitmap, text, or metafile). Then select the location where you want to store the waveform you are viewing.
	Click to zoom in or out. When zoomed in, there is a scroll bar at the bottom of each graph, used to move to the left or right.
	Click to view the display as a waveform plot.
	Click to view the display as a spectral plot. This feature does not work for the "3" or "VI" views.

Button	Description
	Click to view the display as an RMS plot.
	Select any phase; then click here to view the digital status of any of the digital channel(s) that have been set up for this circuit monitor.
	Not currently used.

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 184](#). 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 Citect.chm help file ([I/O Devices Properties](#)).

Tag Status: This screen views the status of the display value, override status, control inhibit status, and field value. You can also change the override status and control inhibit status on this screen.

Changing background colours: Default colours are assigned for the tag override and control inhibit. Change the default background colours in the parameters, not in the ini file.

To change the colour for tag overrides, use `OverrideTextBackgroundColor`. To change the colour for control inhibits, use `ControlInhibitTextBackgroundColor`. For detailed help, see [Page Parameters](#) in the Parameters.chm help.

Use Reports

The reporting feature is an IIS web application that is typically hosted on the same server as the PowerSCADA services. The PLS_Inlude 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.

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 Reporting on page 166](#).

For descriptions of each report type, see [PowerSCADA Expert Reports on page 174](#).

Set up the Runtime Environment for Reporting

Follow these steps to add new items to the project, add the necessary INI parameters for CtAPI and 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 Reports* on page 168.

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 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 the CtAPI Connection for Reporting

Make the following changes to the *Modules.Platform.Citect.dll.config* file:

IPAddress - of the server

UserName - to connect to connect to ctAPI

Password - password to connect to ctAPI

IsEncrypted - set to 1 if you change the password

The first time you run reporting, the password field will be encrypted. *IsEncrypted* will be set to 1. Verify that this has happened before you release the project to a customer.

If you need to modify the username or password, follow these steps:

1. Change the username and/or password in the *Modules.Platform.Citect.dll.config* file.
2. Re-set *IsEncrypted* to 0.
3. Restart Schneider Electric Host (CoreServiceHost) service in the Windows Services Management Console.
4. Connect to reporting (using the menu command you added when you created the report page).
5. Verify that the *password* field (in the *Modules.Platform.Citect.dll.config* file) has been changed from a plain-text password to an alphanumeric encryption string.

Set Up a Display Client for Report Viewing

To be able to properly interact with the 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 Reporting Application

You can use the following tasks within the reporting application to create, view, and email reports:

[Create and View Reports on page 168](#)

[Configure Email Settings to Send Reports on page 170](#)

[Send PowerSCADA Expert Reports via Email on page 171](#)

[Read, Export, Print, and Edit Reports on page 173](#)

Create and View Reports

This section explains how you can create 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 Reporting on page 166](#).

You can generate reports in two ways:

- run a new report by entering parameters
- run a report from a saved configuration

Run a New 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 Reports on page 178](#).

Run a Report and Save its Configuration

To create and save a report configuration:

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 (MultiDeviceReport, SingleDeviceReport, etc.)

<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 Report Using a Saved Configuration

To view a 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 Report Using a Saved Configuration

Use this procedure to modify a configuration that you saved according to "Create a Report and Save its Configuration," above.

Browse to the "show configuration" URL for the report that you want to modify. Use this format:

http://<Server-
Name>/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\PowerSCADA Expert Reporting\Reporting\ReportConfigurations\ (64 bit)

Configure Email Settings to Send Reports

Follow these steps to configure email settings so that you can send PowerSCADA Expert reports to multiple email addresses.

NOTE: You must configure the SMTP server and email list(s) before you email reports. See [Send PowerSCADA Expert Reports via Email on page 171](#) for instructions on sending these emails.

SMTP Server and From Address

You must modify web.config to include the SMTP server to be used when sending email.

Locate the following text in the web.config file:

```
<system.net>
  <mailSettings>
    <smtp deliveryMethod="Network" from="FROM_EMAIL_ADDRESS">
      <network host="INSERT_SMTP_SERVER" port="25"
        defaultCredentials="true" />
    </smtp>
  </mailSettings>
</system.net>
```

Replace the INSERT_SMTP_SERVER with the IP address or fully-qualified name of your SMTP server.

Replace the FROM_EMAIL_ADDRESS with an appropriate email address. (The FROM_EMAIL_ADDRESS only applies when sending email via URL or ReportMailer (CICODE). It has no impact when sending email from the web report viewer.)

If your SMTP server requires authentication, refer to Microsoft's documentation at [http://msdn.microsoft.com/en-us/library/ms164242\(v=vs.100\).aspx](http://msdn.microsoft.com/en-us/library/ms164242(v=vs.100).aspx).

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 theReporting\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 PowerSCADA Expert Reports via Email

Before you can send PowerSCADA Expert reports via email, you must configure the SMTP server and email list(s) before you email reports. See [Configure Email Settings to Send Reports on page 170](#) for instructions on configuring this information.

There are three ways to send PowerSCADA Expert 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.

Web.config Timeout Value: The web.config file also includes a timeout value: <add key="EmailTimeout" value ="600"/>

This is the maximum time (in seconds) to allow for a report to be generated and sent to a list of recipients.

Default = 600 seconds.

(This is only applicable when sending reports via URL or ReportMailer. It has no impact when sending email from the web report viewer.)

To send a report to an existing email list, visit the following URL:

http://<Server-
Name>/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 Reports on page 168](#) 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 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 7.30\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 7.30\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 Reports

You can schedule the emailing of reports by executing the above Cicode as an action from a timed event. See the topic [Configuring Events](#) in the CitectSCADA.chm help file for more information.

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 Reports

After you create a 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.

Option	Description
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 PowerSCADA Expert Reports via Email on page 171 .

Export a Report

Before you can print a report, you must export it into a format that can be printed.

You can export to:

- HTML
- PDF
- BMP
- EMF
- GIF
- JPEG
- TIFF
- PNG
- XML

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 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 Reports

During runtime, you can create, view, save, and print reports.

Before you can view reports, there must be data logged for the project. See [Use the Analysis Page on page 160](#) for help.

You must also create a report tab for the graphics page in runtime mode. See [Set up the Runtime Environment for Reporting on page 166](#).

After you have logged trend information, you can create and view reports. From the graphics page in the runtime environment, click the reports tab and choose the report type you want to create:

[Single Device Usage Reports on page 175](#)

[Multi Device Usage Reports on page 175](#)

[Tabular Reports on page 176](#)

[Trend Reports on page 177](#)

See also:

[Report Localization on page 179](#)

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 Reporting](#).

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 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 Reporting*.

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 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 Reporting*.

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 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 Reporting*.

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 Reports*.

Enable Windows Authentication for Reporting

You may want to use Windows Authentication for logging into the 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 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. For more information: <http://support.microsoft.com>, and search on kb/815179.

URL Routing for Reports

The reporting application uses Asp.NET extensionless URL routing. Depending on your operation 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>

Report Localization

To localize 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 Reporting\bin\Resources directory.

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.

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 (Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 7.30\User\[projectname]. For Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 7.30\User\[projectname]), add the following instructions:

Add the following instructions to the Citect.ini file (changes can be added directly to the Citect.ini file or in the project database, via menu command: System > Parameters in Project Editor):

[MultiMonitors]

Monitors = 4

StartupPage1 =

StartupPage2 =

StartupPage3 =

StartupPage4 =

where

Monitors = n; n = number of monitors used to display the pages in the project

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 does NOT work, because it requires a valid license.
- 15 minutes with a maximum of 50,000 real I/O.
- 10 hours with no static points and a maximum of one dynamic real I/O. This is useful for demonstrations using memory and disk I/O. The product starts in this mode if no static points are configured.

If you want to demonstrate DDE, CTAPI, or ODBC writes to PowerSCADA Expert in this mode, you can only write one point. To write to more than one point, you must force the product to start in 15 minute-50,000 I/O demo mode by creating at least one static I/O point.

For this to work, you must configure a real variable tag, with an accompanying PLC or I/O device. The tag must be used by a page or in Cicode. If you do not have a real I/O device connected, PowerSCADA Expert displays a hardware error, which you can disable using the IODeviceControl function.

- 8 hours with a maximum of 42,000 real I/O. This is only available through special PowerSCADA Expert Integration Partners (CIP) keys.

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 183*](#)

[*Add and Modify User Accounts on page 184*](#)

[*Change Access Rights on page 186*](#)

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 186](#) 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 183](#) 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 (Start > Programs > Schneider Electric > PowerSCADA Expert 7.30 > PowerSCADA Expert PC-based Help).

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 PowerLogic. These genies use a particular naming convention, which is described in the table below. In the PowerLogic library, each genie name begins with “pls,” and is followed by a description of the type of genie according to this table:

first	second	third	fourth
pls indicates PowerLogic library	alarm = alarm	base = primitive genies	1 = small
	ansi = ANSI style	cb = circuit breaker	2 = large
	display = equipment details	sw = switchgear	
	gen = generic	cmd = control genies	
	iec = IEC style	eq = equipment (devices)	
	style = navigation		

Additionally, the actual genies have abbreviated names. When you highlight a genie name, the abbreviation displays above the top row of genie icons.

The following tables list and define the individual genies in each of the PowerLogic libraries.

PLS_ALARM

This library includes genies that provide functionality to alarm displays.

Genie Abbreviation	Description
Field	data portion of an alarm row
Row	a genie with a collection of fields
Selector	for column resizing
Setpoint	a setpoint row

PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2

These libraries include a variety of base symbols used to created genies for ANSI equipment.

1 = small size

2 = large size

Genie Abbreviation	Description
sl_battery_gen	single-cell battery
sl_battery_multi	multiple-cell battery
sl_capacitor	capacitor
sl_capacitor_vari	variable capacitor
SL_Closed_HV	closed circuit breaker position
sl_closed_knife	closed knife switch
sl_closed_lv	closed low-voltage circuit breaker
sl_conductive_path_1	conductive path 1

Genie Abbreviation	Description
sl_conductive_path_2	conductive path 2
sl_conductive_path_3	conductive path 3
sl_conductor_junction	conductor junction
sl_contact_nc_closed	contact break, closed
sl_contact_nc_open	contact break, open
sl_contact_no_closed	contact make, closed
sl_contact_no_open	contact make, open
sl_contact_term	contact terminal
sl_ct	current transformer
sl_in_cb_rack	incoming, circuit breaker racked out, plug
SL_In_Rack	incoming, circuit breaker racked out, socket
sl_in_sw_head	incoming switch head
sl_inductor	inductor
sl_inductor_adjust	adjustable inductor
sl_inductor_gen	general inductor
sl_inductor_magcore	magnetic core inductor
sl_inductor_vari	variable inductor
sl_open	open symbol
sl_open_fuse_sw	open isolating fuse-switch
SL_Open_HV	open high-voltage circuit breaker
sl_open_knife	open knife-type switch
SL_Open_LV	open low-voltage circuit breaker
sl_out_cb_rack	outgoing, circuit breaker racked out, plug
SL_Out_Rack	outgoing, circuit breaker racked out, socket
sl_pb_break	push-button, break
sl_pb_make	push-button, make
sl_pb_term	push-button, terminal
sl_pt	potential transformer
sl_relay	relay
sl_resistor	resistor
sl_resistor_adjust	adjustable resistor
sl_resistor_vari	variable resistor
sl_separable_con_closed	separable connector, closed
sl_separable_con_open	separable connector, open
sl_separable_con_plug	separable connector plug
sl_separable_con_socket	separable connector socket

PLS_ANSI_CB_1 / PLS_ANSI_CB_2

These libraries include genies for ANSI-type high-voltage and low-voltage drawout circuit breakers.

1 = small size

2 = large size

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
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)

Genie Abbreviation	Description
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

Genie Abbreviation	Description
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
transformer_2_in_Y	transformer 2: star (wye), on-line
transformer_2_out	transformer 2: general, off-line
transformer_2_out_D	transformer 2: delta, off-line
transformer_2_out_Y	transformer 2: star (wye), off-line (no 2 IN D? or 1 IN D?)

PLS_GEN_CMD_1 / PLS_GEN_CMD_2

These libraries include genies that control display of popups and values:

1 = small size

2 = large size

Genie Abbreviation	Description
CmdDetail	provides access to the equipment detail popup
cmddetail_meter	provides access to the meter detail popup
Control	control in a circuit breaker

Genie Abbreviation	Description
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

Genie Abbreviation	Description
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
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 Mircologic meters
power_meter	PowerSCADA Expert power meter
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 197](#)

[Performance Tuning Parameters on page 205](#)

[Waveform Parameters on page 213](#)

[Sepam Event Reading Parameters on page 215](#)

[MicroLogic Modules Configuration Parameters on page 216](#)

[Data Replication Parameters on page 217](#)

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

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

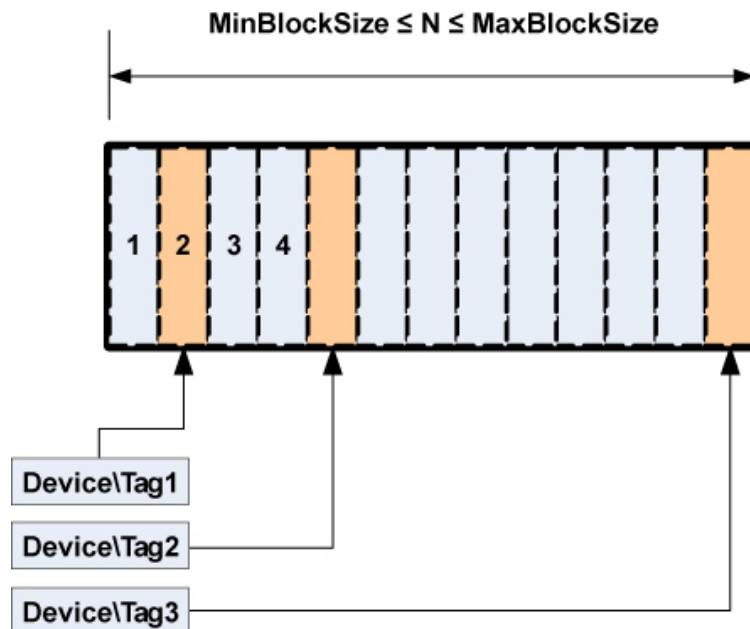
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_B]

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 500T

Advanced Tag Block Capabilities (Invalid Memory Access Blocks defined)

Some devices may restrict access to certain memory registers. Such registers may be available for read only, write only or may not be available at all, resulting in a MODBUS exception when the registers are addressed.

Definition: Blocks of registers that cannot be read or written to are referred to 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.

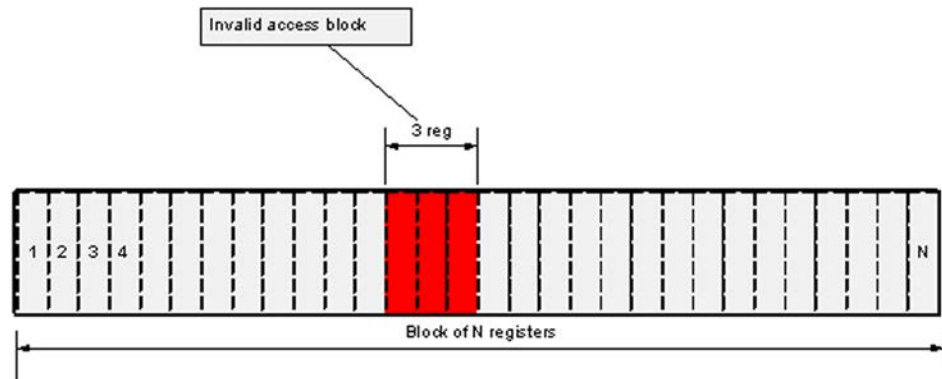
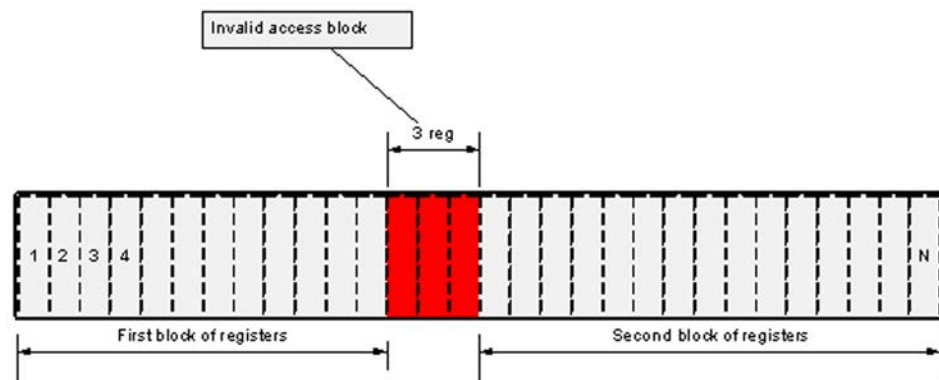


Figure 1 shows how the "invalid memory access block" affects MODBUS register blocking. In this situation, if the driver does not know that the block of 3 inaccessible registers exists, it will try to block all registers from 1 to N (depending on data that was requested by the real-time data collector). 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

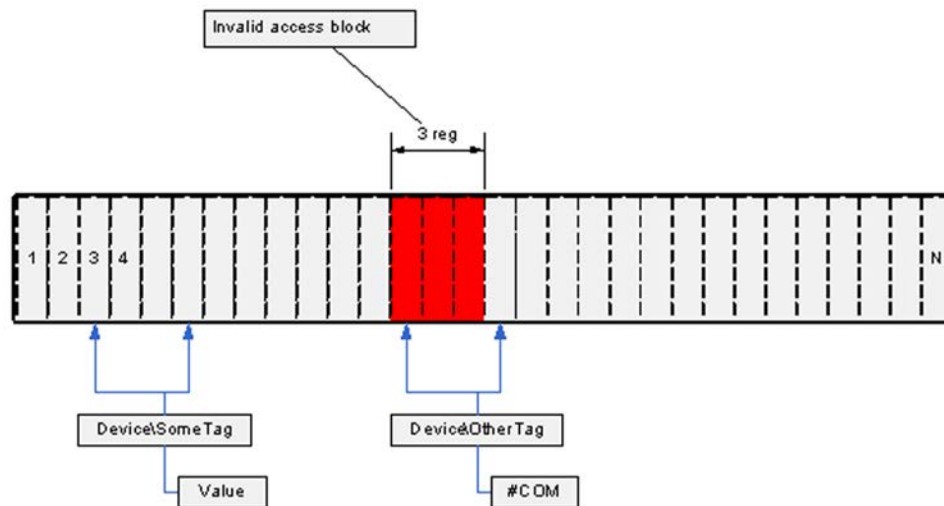
Example:

```
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\Sw1Str 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

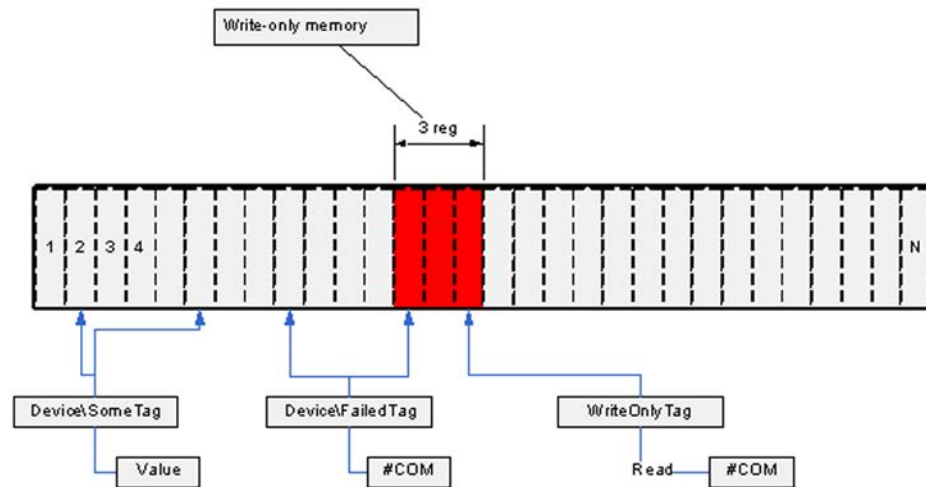
[ERROR] [MISC] [RealTimeData::BlockBuilder::AddDataPoint()] Cannot
add datapoint, one or more invalid memory addresses fall into non-
splittable block

[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.

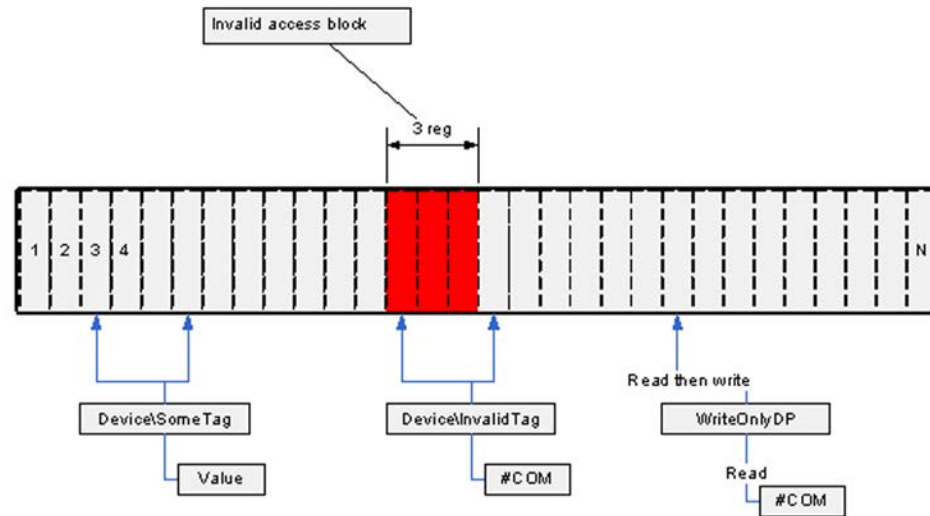
Write-only Memory

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.

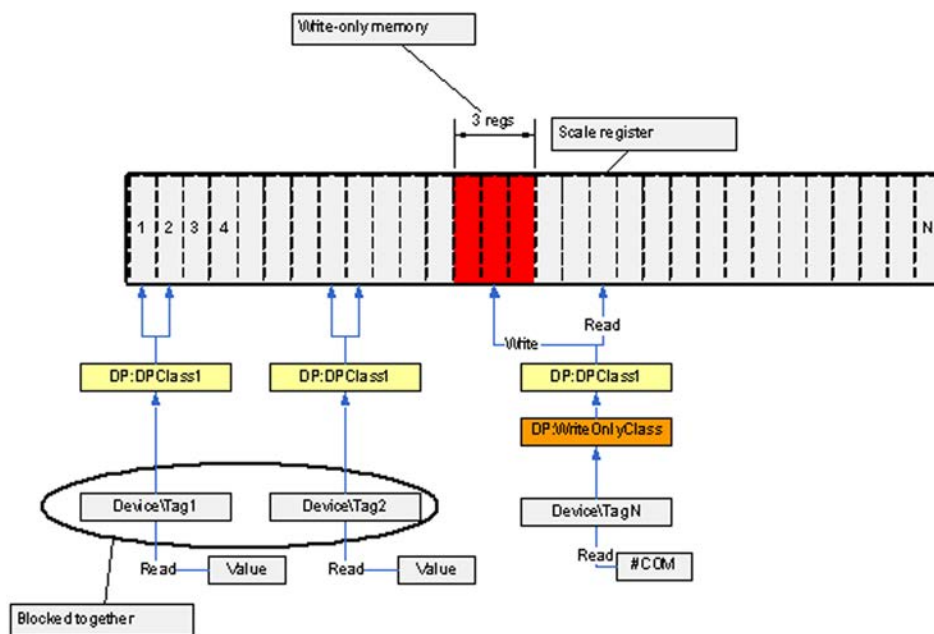
Tags found invalid, due to intersection with invalid memory areas, are not blocked with “good” real-time tags and will not therefore interfere with them.

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 (clusters, individual devices, etc.)

There is also a tag that will change an individual device's setting at runtime (it will reset to the default when you restart the project). This tag is LLNO\WaveformCollectionEnabled. 1 = True, 0 = False.

WaveformMatchMargin

Alarms are matched to waveforms by the timestamp of each. This parameter is the maximum difference between alarm timestamp and waveform timestamp for 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.

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 7.30\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

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 181](#) 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 77](#) 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 183](#) 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.

Can I install ION E or SMS on the same computer as PowerSCADA Expert?

We do not recommend this practice. Each of those applications uses SQL server, which requires so much of the CPU that system performance would be severely degraded.

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 69](#) 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, etc.

- For example, Site_Building_Panel_device would be named Sx_Bx_Px_Device. (Site1_Building1_Panel1_CM41 — S1_B1_P1_CM41).
- The fewer levels you have, the more characters you can have in each level.

Device Comment:

Use this field for an alias of the device name.

This comment will be placed in the Equipment database, which is accessible from Cicode.

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 PowerLoic SCADA, which will cause a communication loss. Enabling the "retry on exception" will re-try the request.

How do I use MODBUS communications methods?

MODBUS TCP/IP via Gateway: Use this for any device that is not speaking TCP/IP natively. These devices connect through a gateway device such as an EGX or ECC.

MODBUS 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 183](#). For additional information, see [Using_Security.html](#) in the CitectSCADA.chm help file v7.30.

How do I manage projects in the Citect Explorer environment of PowerSCADA 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:

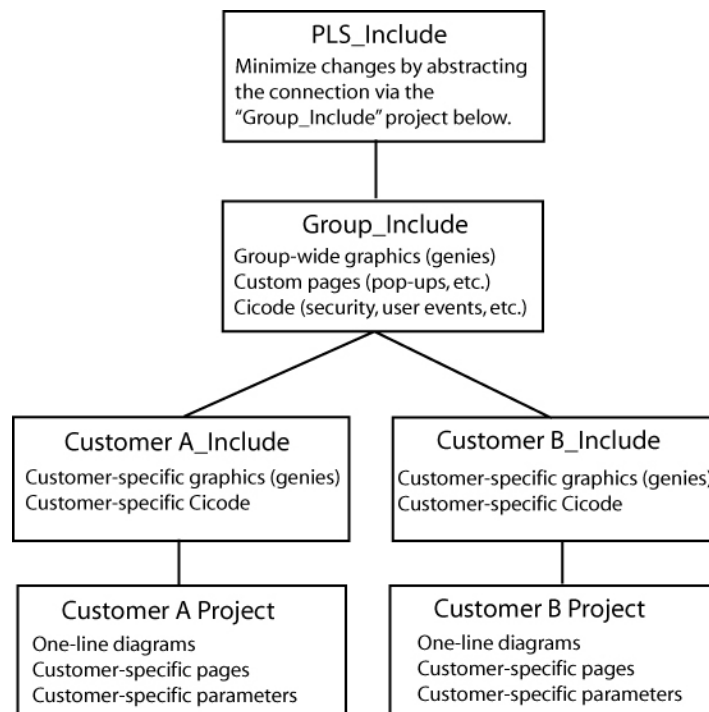
1. Keep original 'Master' copies of the PLS_Example and the PLS_Include projects for reference.
2. The Services group may develop a group-wide "include" project that will act as a conduit between the PLS_Include project and all customer projects (e.g., "Group_Include"). This will make the upgrading of 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 218](#))

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 51](#), “Alarm On Text” and “Alarm Off Text”).

What can I modify during runtime?

Set-point Configuration during runtime: In the Alarm Server entry in the project editor, view the advanced properties (F2) and set publish alarm properties = TRUE

Events Accessible at runtime: By default, 5000 events are accessible (view, filter, print) during runtime. Additional events are logged to disk. The number of events 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 116](#) 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_CM41MMXU1\PhV\zavg > 120).

If the expression is TRUE, the ACTIVE colour will be shown. The active colour is determined by the voltage level assigned.

If the expression is FALSE, the DE-ENERGIZED colour will be shown.

Hard-Coded:

If no upstream devices are available (in the event of an incomer, for example), you may have no other choice than to 'hard code' this field to a '1'. This forces the

busbar to always be ACTIVE.

What INI parameters should I use for debugging?

We recommend that you contact Technical Support before performing any debugging.

Parameter: [PowerLogicCore]

DebugCategory = Error

DebugLevel = All

LogArchive = 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 PowerLogicSCADA checks devices for history files and flushes logging data to disk.

Default: 5000

Range: 1000–3600000 milliseconds.

Miscellaneous Parameters

Parameter: [Kernel]

Task = 20000

Parameter Details:

Kernel Task is the number of tasks. Increasing the number of kernel tasks is used when "Out of Kernel Task" message is received. The change will be likely for large systems.

Default Value: 256

Range: 50–32767

Parameter: [Page]

ScanTime = 250

Parameter Details:

Page ScanTime determines how often the Animator refreshes a graphics page at runtime.

Default: 250

Range: 1–60000 milliseconds

Parameter: [ALARM]

ScanTime = 500

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 205](#).) 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 205](#).) 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 genres 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 includes 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 205](#) 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 [General PowerSCADA Expert Parameters on page 197](#) for details)

Index of Terms

1

- 1.#R
 - value returned in instant trend 161

A

- access rights
 - adding for users 51
 - changing for users 186
 - Windows integrated users 184
- ACE969TP device type
 - used with DNP3 61
- ACE969TP module
 - used with DNP3 61
- additional sources of information 20
- address
 - correcting missing or invalid 37
 - functional 38, 64
 - defined 224
- advanced tag block capabilities
 - configuration notes 209
 - invalid block tag definition syntax 209
 - tag blocking notes 211
 - write-only memory 210
 - write-only tags 211
- AdvOneLine.csv 122
- AdvOneLine.exe 122
 - stopping and starting 124
- AdvOneLine.ini 122
- AdvOneLine.ini settings
 - one-lines 123
- AdvOneLineDebug.Bus.csv 122
- AdvOneLineStatusLog.txt 122
- alarm
 - adding an onboard tag 143
 - associated with waveforms 151
 - category 159
 - changing pickup/dropout text 237
 - changing the severity 141
 - codes
 - unique identifier 48
 - filtering
 - custom 159
 - historical logging parameters 142
 - issues 157
 - level
 - setting in Profile Editor 52
 - onboard 140
 - pc-based 140
 - setting up in Profile Editor 53
 - priority 160
 - setting up based on an enumeration 141
 - states
 - startup parameters 218
 - tags
 - adding 140
- alarm filters 159
- alarm format code definitions 78

- alarm log 156
 - column headings described 158
 - filtering 155-156, 159
 - freezing 155
 - logging data to disk 156
 - printing 157
 - problem with alarms not displaying 142, 157
 - sorting 155-156
- Alarm On/Alarm Off text 54
- alarm pages
 - adding to the project 140
- Alarms/Events page
 - equipment column 155
 - filtering alarms 160
 - remove/insert/move columns 155
 - viewing 155
- AlarmSummaryLength parameter
 - maximum number of entries in the Event Log 142
- analysis page 160
- animation
 - loss of communication not filtering correct 155
 - not display correctly when genie dimensions
 - wrong 126
- animation numbers (ANs)
 - used in objects 236
- array
 - genie properties 129
- assigning tags to generic I/O points 41
- assumptions
 - about PowerLogic SCADA system integrators 11
 - before system integrator can set up system 11
- ATS
 - configuring 125
 - errors 134
 - information 134
- audible alarms
 - setup 143
- auto assign genie 130, 132
- auto configure genie 130, 132
- automation interface
 - "out of memory" message 103
 - adding multiple devices 103
 - required information 104

B

- back ground colour
 - changing in projects 115
- background colours
 - changing 165
- backup
 - Citect project file 144
 - Profile Editor project file 68
- base unit and conversion
 - adding and editing 34
 - deleting 35
- before you use the Profile Wizard 89
- block writes
 - using 86
- BRCB
 - entering for IEC 61850 device 100
 - entering for IEC61850 device 100

- breaker
 - configuring 126
 - errors 136
 - information 135
- browser navigation buttons do not work 116
- busbar
 - configuring 125
 - creating and configuring 238
 - errors 135
 - information 135
- busbars
 - two with same line active 133

C

- CacheRefreshTime parameter 240
- calculating disk storage 148, 160
- CASE
 - changing user access rights 186
- category/subcategory
 - adding 42
 - deleting 43
 - editing 42
 - managing 232
- changing alarm severity 141
- circuit breaker status
 - using special tags to control 76
- Citect.ini
 - used to customize a project 145
- citect.ini file
 - setting up multiple monitors 180
- Citect.ini settings
 - one-line 123-124
- cluster
 - adding 89
- CLUSTER name (%CLUSTER%)
 - changing 242
- cluster setup
 - in the Profile Wizard 93, 95, 97, 99
 - redundant I/O servers 91
- colour
 - changing background in projects 115
 - changing for genies 115
- colours
 - assigning to one-lines 122
 - changing background 165
- comment field
 - used in translation 90
- comms loss
 - troubleshooting 244
- communications loss 155
 - troubleshooting 244
- compile and run 17
- Compile Errors window
 - viewing PowerLogic SCADA errors 117
- composite device type
 - creating 58
- Computer Setup Wizard
 - running after first compile 17, 89
 - when adding a startup page 116
- configuration tools
 - described 12

- control format codes 78
- control inhibit 165
- controls
 - how they operate 37, 49
- conversions
 - to base units 34
- CoreServiceHost
 - restart when changing runtime projects 166
- creating
 - CSV file 41
 - device profile 51
 - engineering template 32
 - genie 128
 - one-line on a graphics page 116
 - PowerLogic SCADA project 114
 - project 62
- CSV file
 - creating, viewing, printing 31, 41
- CtAPI connection
 - reporting 167
- CTZ file
 - backup file 144
- custom
 - control format codes 79
 - filters
 - alarms 159
 - genie
 - creating 128
 - icon 30
 - reset
 - adding 48
 - symbol
 - creating 129
 - tag
 - deleting 46
 - deleting when associated with a device type 46
 - editing 46
- custom tag
 - adding 44
- customer support contact information 20
- customize tag names
 - sub-tab 62
- customizing a project with Citect.ini 145
- customizing tag names 63, 65

D

- data points
 - defining individually in an address 38
- data replication parameter 217
- datasets
 - adding/editing 55
- DBF Add-In toolbar will not install 244
- debugging
 - INI parameters to use 239
- define
 - device types and tags 36
 - enumeration 76
- demo mode
 - using 181
- design-time environment
 - described 113

- device
 - adding multiple 234
 - adding to the PowerLogic SCADA project 93
 - name
 - requirements 233
 - popup
 - overlapping fields 241
 - profile
 - adding 64
 - creating 51
 - defined 16
 - deleting 54
 - editing 54
 - removing from a PowerLogic SCADA project 102
 - tags
 - defining 36
 - type
 - adding, editing, deleting 38
 - defined 16
 - defining 36
 - deleting 40
 - editing 40
 - locking 30
 - unlocking 30
- device communications issues
 - troubleshooting 233
- device profile
 - add as default 64
- device type
 - category/subcategory
 - adding 42
 - deleting 43
 - editing 42
 - composite 58
 - drivers
 - INI parameters 146
 - drivers and protocols
 - list of supported 12
 - name
 - limitations 39
- Device Wizard
 - before you use it 89
- devices in the PowerLogic SCADA project
 - editing 101
- disabling "tag not exist" prompt. 116
- disk storage calculations 148, 160
- display client
 - report viewing 167
- display projects button
 - adding a project in the Profile Editor 16, 64
- DNP3-TCP device
 - adding 97
- DNP3 protocol support 61
- DNP3_serial
 - adding to PowerLogic SCADA project 95
- documentation in this release 20
- driver parameters
 - editing 13
 - selecting for a profile 59
 - setting up in the Profile Editor 54
- drivers
 - choosing for device types 39

- generic power device 12
- how do they work? 13
- subscription expirations 13
- subscription types 13
- duplicate devices
 - defined 13

E

- email
 - configure to send reports 170
 - sending reports 171
- encryption
 - password 121, 124, 167
- engineering template
 - creating 32
- engineering units
 - adding 34
 - exceptions 33
 - adding from the Project tab 64
 - deleting 35
 - editing 35
 - multipliers and offsets 33, 35
- English.dbf
 - used in translation 94, 96, 98, 100
- English.dbf file
 - used for translation 94, 96, 98-99
- entering copy for translation 92, 94, 96, 98-99
- enumeration
 - defining 76
 - setting up alarm based on 141
 - status enumeration logic code 82
- equipment column
 - alarms/events pages 155
- equipment name
 - adding via Automation Interface 104
 - adding via Profile Wizard 94, 96-97, 99
 - using when creating a one-line 116
- equipment pop-up page 161
- Equipment.Profiles
 - file storage location 67
 - viewing 63
- errors
 - ATS 134
 - breaker 136
 - busbar 135
 - meter 137
 - sources 137
 - switch 134, 136
 - transformer 138
- errors and warnings
 - correcting in the graphics page 117
 - one-line 131
- event filters 159
- event log 156
 - acknowledge, silence, print 157
 - logging data to disk 156
 - problem with alarms not displaying 142
- events
 - historical logging 142
 - storage limitations 142

Example Automation.xls
 using to export multiple devices 103
Excel DBF Add-In toolbar will not install 244
expanding PowerSCADA Expert screens
 F2 key 90
export
 PowerLogic SCADA 66
 Profile Editor (backup) 68
 project files from Profile Editor 66
 SCL files 68

F

F2 key
 use to expand PowerLogic SCADA screens 90
 use to expand screens 90
FAQs 230
 "out of memory" message in Automation
 Interface 243
 #COM error for circuit breaker status 244
 adding more than one device at a time 234
 adding/removing devices in Profile Wizard 232
 auditing user logins/logouts in the Event Log 243
 browser navigation buttons do not work 238
 can I change %CLUSTER% name 242
 changing default pickup/dropout text 237
 creating and configuring busbars 238
 creating device profiles without PowerLogic
 drivers 230
 creating users in Runtime 234
 device name requirements 233
 device type vs device profile 232
 device writing to its registers 242
 Excel DBF Add-In toolbar will not install 244
 How do I troubleshoot device communications
 issues? 233
 how to use MODBUS communications methods 234
 if tag is defined, is it polled 241
 INI parameters used for debugging 239
 ION E or SMS installed with PowerLogic SCADA 232
 licensing error at web client 242
 managing categories and subcategories 232
 managing projects in PowerLogic SCADA 235
 naming conventions for servers/clusters 234
 overlapping fields in device popup 241
 performance tuning parameters 240
 preventing reads/writes to invalid areas of memory 243
 synchronize new PC with master PC 232
 what are genies 236
 what can be set up in logging and archiving 238
 what can I modify during runtime? 237
 working with templates 236
FileFormat parameter
 historical logging of alarms and events 143, 157
filtering genie states 131, 158
filtering the alarm and event logs 159
filters
 real time and alarm 45
First dbf record
 error 244

force edit 71
 when importing 71
format codes
 alarm definitions 78
 control definitions 78
 custom control 79
 definitions 77
 predefined control 78
 real-time definitions 77
 reset 79
frm file
 associated with new genie 236
FRM file
 formatting genie appearance 128-129
functional address 38, 64
 definition 224
 edit 43

G

G3200 setup 60
generic I/O points
 assigning tags 41
generic MODBUS drivers
 included in system 12
genie
 adding to a drawing 116
 auto assign 130, 132
 auto configure 130, 132
 busbar 125
 configuration
 viewing 130
 creating and editing 128
 grouping 131
 icon 116
 in Graphics page 236
 libraries
 ITEM1 194
 PLS_ALARM 187
 PLS_ANSI_BASE_1/PLS_ANSI_BASE_2 187
 PLS_ANSI_CB_1/PLS_ANSI_CB_2 188
 PLS_ANSI_SW_1/PLS_ANSI_SW_2 190
 PLS_DISPLAY 190
 PLS_GEN_BASE_1/PLS_GEN_BASE_2 190
 PLS_GEN_CMD_1/PLS_GEN_CMD_2 191
 PLS_GEN_EQ_1/PLS_GEN_EQ_2 192
 PLS_IEC_BASE_1/PLS_IEC_BASE_2 192
 PLS_IEC_CB_1/PLS_IEC_CB_2 193
 PLS_IEC_SW_1/PLS_IEC_SW_2 193
 PLS_METER 194
 meter 126
 repairing 132
 resizing 126
 source 127
 state filters 131
 switch/circuit breaker 126
 transformer 127
genie colour
 changing in project pages 115
GenieConfiguration.xml file
 one-line 129
getting more information 20

- graphics library parameters 218
- graphics page
 - adding in PowerLogic SCADA 115
 - setting as startup page 116
- Graphics runtime environment
 - editing appearance 117
- group by type or page 131

H

- help files 20
- historical parameters of alarms
 - setup 142
- historical trends
 - viewing 160
 - viewing from a one-line 164

I

- I/O server
 - adding before using the Equipment Wizard 89
- ICD file
 - editing 68
- icons
 - locked and custom 30
- IEC 60870-5-104
 - support 61
- IEC 61850
 - adding device to PLS project 99
 - datasets
 - adding/editing 55
 - general workflow 19
 - report control blocks
 - adding/editing 57
- IEC 61850 tags
 - construction 75
 - logic codes described 79
- IED
 - defined 225
- IID
 - defined 225
- import
 - files into Profile Editor 69
 - import reconciliation screen
 - merge and match 71
 - match option 72
 - merge option 72
 - project files into Profile Editor 66
 - removing import templates 74
 - SCL files 73
 - template
 - applying 69
 - using 73
- import Import reconciliation screen
 - search 70
- INI parameters 195
 - for historical reports 166
 - for supported device type drivers 146
 - to use in debugging 239
- instant trend
 - pens stop updating 161
 - viewing 161

- Internet Explorer 9 emulation
 - reporting 167
- intervals
 - trend tags 59
- invalid or missing tag address
 - correction 37
- IP address
 - entering in Example Automation.xls 104, 108
- Is Device Tag
 - reporting for composite devices 53

J

- JBus
 - adding 12, 39

L

- launching the profile editor 30
- licensing error at web client 242
- locked device type
 - cannot delete 40
 - unlocking 30
- locked icon 30
- locking a device type 30
- logic codes
 - definitions 79
- loopback address
 - used for network addresses 89-90

M

- main menu options 31
- manuals 20
- memory device
 - one-lines 122
- menu
 - main options 31
- Menu Configuration
 - used to edit Graphics page menu information 17, 117
- Menu Configuration tool 117
- metadatafields.xml
 - use to edit alarm filters 160
- meter
 - configuring 126
 - errors 137
 - information 136
- MicroLogic
 - genie in Graphics Builder 194
- MicroLogic P
 - #COM error on genie status page 162, 244
- Microsoft Excel
 - using to add devices to the database 103
- migration utility 21-22
- MODBUS communications methods 234
- multi-device reports 175
- multiple monitor support 180

N

- naming ports 100
- navigation arrows
 - runtime environment 154

- network address
 - adding 90
- network address and servers
 - adding before using Profile Wizard 89
- NetworkTagsDev I/O device
 - adding 91
- normally closed
 - control tags 37

O

- OLE automation interface
 - add and remove devices 105
- onboard alarm tag
 - adding 143
- onboard alarms 140
- one-line
 - adding to a graphics page 116
 - colouring 122
 - communication errors 131
 - PowerSCADA Expert
 - described 121
 - repairing 132
- one-line animation file locations 122
- One-Line Configuration Utility 130
- one-line device 122
- one-line flowchart 120
- one-line memory device 122
- out of memory message
 - when running Automation Interface 103
- override
 - tag name 53, 58
- override tag status 165

P

- pack
 - after deleting a cluster 91
 - using to delete data 119
- Pagemenu.dbf
 - editing 117
- parameters
 - advanced tag block capabilities 208
 - alarm and event state 218
 - AlarmFormat 159
 - AlarmSummaryLength 142
 - AlarmUseConfigLimits 141
 - BandwidthAllocation (port level) 205
 - BandwidthAllocation (device level) 205
 - cacheRefreshTime 200
 - CCM (Micrologic) 216
 - Citect INI 195
 - ClientAutoLoginMode
 - used to set up Windows integrated user 184
 - CommandsBandwidth 205
 - data replication 217
 - database described 195
 - driver
 - editing 13
 - selecting for a profile 59
 - enableScatteredReads 206
 - EventBandwidth 205

- EventIdle 215
- EventTable 215
- graphics library 218
- HighScanRate 207
- initCacheTimeout 200
- initUnitCheckTime 200
- kernelStatisticUpdateRate 197
- LowScanRate 207
- maxBlockSize 207
- Micrologic 216
- minBlockSize 207
- ModbusBase 202
- module-specific packet blocking optimisation
 - settings 216
- packet blocking optimization 205
- percentBlockFill 206
- performance tuning 205, 240
- PLSEventLog 142
- PLSEventLogFileFormat 157
- PowerLogic INI driver 197
- RegMode 202
- retry 198
- retryexception 198
- retrytimeout 198
- Sepam Event Reading 215
- standbyCheckTime 199
- standbyRefreshRate 199
- statusRegister 201
- statusregistertype 201
- statusUnitCheckTime 199
- Tag Scan Rate 207
- time synchronization 146
- time zone settings 146
- TimeBandwidth 205
- timeout 197
- timeZone 202
- usewritemultiregistersonly 197
- watchtime 197
- waveform 213
- WaveformBandwidth 205
- WaveformCheckTime 213
- WaveformMatchMargin 213
- WaveformsDisable 213
- WaveformZone 214
- password encryption 121, 124
 - reports 167
- pc-based alarms 140
 - setting up 53
- performance tuning parameters 205, 240
- pickup/dropout text
 - changing for alarms 237
- PLS export 66
- PLS_Example project 17, 113
- PLS_ReportDsp()
 - Cicode function
 - used in report configuration 166
- PLS_ReportPage
 - used in report definition 166
- PLSAnimBusbar()
 - Cicode function for custom genie animation 129

- PLSEventLog parameter
 - allowable values 157
 - historical logging of events 142
 - PMCU
 - adding onboard alarms and waveform captures 141
 - defined 225
 - polling rules for tags 241
 - port renaming
 - serial 95
 - ports
 - naming 100
 - renaming
 - serial 96
 - TCP 95, 98
 - PowerLogic Driver Parameters 197
 - PowerSCADA Expert
 - backup 144
 - described 11
 - naming conventions 234
 - project
 - creating 114
 - uninstall and reinstall 15
 - PowerSCADA Expert one-lines
 - described 121
 - error logging 132
 - GenieConfiguration.xml file 129
 - introduction 130
 - SupportedGenies.xml file 129
 - predefined control format codes 78
 - predefined project
 - PLS_Example 113
 - printing
 - CSV file 41
 - printing the alarm log 157
 - privileges
 - user
 - PowerLogic SCADA 183
 - profile editor
 - export 68
 - launching 30
 - overview 25
 - Profile Editor
 - importing files into 69
 - Profile Editor support utility 61
 - Profile Editor.exe.config
 - using to change trend tag scan intervals 147
 - Profile Wizard
 - adding a device 89
 - adding a device to a project 93
 - before you use it 89
 - editing device information in a PowerLogic SCADA
 - project 101
 - removing a device 102
 - profiles
 - adding, editing, deleting 52
 - enabling waveforms 51
 - file storage location 67
 - name limitations 53
 - viewing 51
 - project
 - adding 64
 - cannot add 244
 - creating in PowerLogic SCADA database 66
 - deleting 65
 - editing 65
 - exporting (to Citect from Profile Editor) 66
 - name limitations 64
 - problems when renaming in Citect 114
 - reusing 114
 - setting up 62
 - project parameters
 - sub-tab 62
 - using to add information for imports 63
 - Project Parameters sub-tab 65
 - project XML files
 - storage location 67
 - projects
 - managing in PowerLogic SCADA 235
- ## Q
- Quantum PLC
 - genie in Graphics Builder 194
 - time-stamped events 220
 - quick start 16
- ## R
- rack position
 - displaying incorrectly 126
 - read logic codes 79
 - read/write logic codes 86
 - real-time format codes definitions 77
 - real-time lists
 - viewing from genies 165
 - redundant I/O servers
 - cluster setup 91
 - refreshing tags in the Project tab 63, 67
 - repair
 - one lines 132
 - project 131
 - upgrade project 131
 - ReportMailer utility
 - sending email reports 172
 - reports
 - accessing, creating, modifying, removing 168
 - adding to runtime menu 166
 - configure email to send 170
 - CtAPI connection 167
 - editing appearance 174
 - enable Windows Authentication 177
 - export 174
 - exporting, printing, saving 173
 - feature described 166
 - INI parameters 166
 - Internet Explorer 9 emulation 167
 - introduction 174
 - localization 179
 - multi device usage 175
 - password encryption 167
 - PLS_ReportDsp() 166
 - PLS_ReportPage 166
 - sending via email 171
 - set up a display client for report viewing 167

- setting up in the runtime menu 166
- single device usage 175
- SMTP server and from address 170
- tabular 176
- trend 177
- URL routing 178
- using ReportMailer 172
- reserved names
 - do not use in projects 227
- reset format codes 79
- resets
 - adding custom 48
 - how they operate 37, 49
- restore
 - project file 144
- resx files
 - localization 179
- reusing a project file 67, 114
- roles
 - adding user 184
- runtime environment
 - described 12
 - launching and closing 153
 - viewing 154

S

- SCADA
 - defined 11
- scan intervals
 - trend tags 147
- scattered reads
 - using enableScatteredReads parameter 206
- SCL
 - defined 227
 - export 68
- SCL files
 - importing 73
- screen resolution
 - Profile Editor 30
- Selected Device Profiles
 - sub-tab 62
- selecting 34
- Sepam
 - genie in Graphics Builder 194
- serial device
 - adding to PowerLogic SCADA project 95
- servers
 - adding 90
 - adding to PowerLogic SCADA 89
 - described 89
- servers/clusters
 - naming conventions 234
- setpoints and delays
 - entering 140
 - entering from the runtime environment 164
- setting up
 - alarm based on an enumeration 141
 - audible alarms 143
 - device type categories 42
 - normally closed/normally open controls 37

- severity
 - changing for alarms 141
- show by type or page 131
- sim source
 - transformer top and bottom 138
- single device reports 175
- SMTP server
 - for reports 170
- sorting the alarm log 52, 155-156
- source
 - configuring 127
 - errors 137
 - information 137
- start and stop AdvOneLine 124
- startup page
 - setting 116
- state filters 131
- storage limitations
 - calculating disk space 148, 160
- subscriptions
 - drivers 13
- support
 - technical 20
- SupportedGenies.xml file
 - PowerSCADA Expert one-lines 129
- switch
 - configuring 126
 - errors 136
 - information 134-135
- symbol
 - creating custom for custom genie 129
- synchronize new PC with master PC 232
- system creation steps 16

T

- tabular usage reports 176
- tag
 - custom
 - adding 44
 - customizing the name 63
 - IEC 61850 described 75
 - refreshing in the Project tab 63
- tag-polling rules 241
- tag addresses
 - adding 50
 - control 49
 - correcting missing or invalid 37
 - editing 46, 50
 - generic 50
 - onboard alarm 48
 - real-time 47
 - reset 48
- tag definitions
 - printing the CSV file 41
- tag names
 - customizing 65
- tag override 53, 58
- tag status
 - overriding 165
- tag types 75

- tags
 - assigning to generic I/O points 41
 - scanning intervals for trend tags 59
- TCP device
 - adding to PowerLogic SCADA project 95, 98, 100
 - adding to project 93
- technical support 20
- templates
 - creating 32
 - deleting 34
 - imports 73
 - working with 236
- The_Profile_Wizard 88
- time-stamped events
 - Quantum PLC 220
- time synchronization parameter 146
- time zone setting parameter 146
- toolbar buttons
 - waveforms 163
- transfer switch
 - configuring 125
- transformer
 - configuring 127
 - errors 138
 - information 138
 - top and bottom 138
 - voltage-level and pass-through 123, 128, 139
- translation
 - using description fields 92
- trend tag intervals 160
 - changing in the Profile Editor 55
- trend tag scan intervals 59, 147
- trend usage reports 177
- trending (Analysis Page)
 - described 160-161
- trends
 - maximum number of pens 160
 - setting up in Profile Editor 53, 55
 - viewing from within a one-line 164
- typical workflow
 - creating a graphics page 112
 - creating a project in Profile Editor 62
 - creating a system 16
 - Profile Editor 26

U

- uninstalling and reinstalling PowerSCADA Expert 15
- unique ID
 - determining for onboard alarms 48, 143
- units (engineering) and conversions
 - creating or editing a units template 32-33
 - deleting 35
 - deleting a unit template 34
 - editing 35
 - exceptions 33
- URCB
 - entering for IEC 61850 device 100
- URL
 - visit to send reports via email 171
- URL reporting for reports 178

- UsePLSFilter
 - alarm/event filter parameter 159, 204
- user access
 - adding and modifying 182
 - changing rights 186
 - Windows integrated 184
- useraccess
 - default settings 183
- users
 - creating in Runtime environment 234
- using block writes 86

V

- value type
 - controlling real-time display 162
- Variable tags not found
 - disabling the prompt 116

W

- waveform
 - why can't I acquire? 244
- waveform parameters 213
- waveforms
 - enabling in Create Device Profiles tab 51, 63
 - enabling onboard 141
 - linking to alarms 151
 - management 149
 - special digital tags 151
 - storage rules 150
 - toolbar buttons 163
 - viewing from the Alarm Log 156
 - viewing from the runtime environment 156
 - viewing in the runtime environment 162
- web.config
 - securing a URL for sending reports via email 171
 - specifying Windows Authentication, adding roles 178
 - timeout value 171
- Windows Authentication
 - enabling for reports 177
- Windows integrated user accounts 184
- Windows security
 - using to remove user access rights 183
- write logic codes 85
- writes (logic codes) 86

X

- XML files
 - storage for project files 67

Z

- zOL
 - memory device for one-lines 122

Schneider Electric
295 Tech Park Drive, Suite 100
LaVergne, TN 37086 USA

For technical support:
Global-PMC-Tech-support@schneider-electric.com
(00) + 1 250 544 3010
Contact your local Schneider Electric sales
representative for assistance or go to
www.schneider-electric.com

PowerLogic and Citect are trademarks of Schneider Electric. Other trademarks are the property of their respective owners.

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

63220-100-200F1 01/2013
Replaces 63220-100-200E1, dated 06/2011
© Schneider Electric All Rights Reserved.