

# StruxureWare™

## PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards Help Manual

7EN02-0403-00

03/2017



 StruxureWare™

**Schneider**  
Electric



# Safety information

---

## Important information

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



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



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

### **DANGER**

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

### **WARNING**

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

### **CAUTION**

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

### **NOTICE**

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

## Please note

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

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

## Safety precautions

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

### WARNING

#### UNINTENDED EQUIPMENT OPERATION

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

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

### WARNING

#### INACCURATE DATA RESULTS

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

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

### WARNING

#### POTENTIAL COMPROMISE OF SYSTEM AVAILABILITY, INTEGRITY, AND CONFIDENTIALITY

- Change default passwords to help prevent unauthorized access to device settings and information.
- Disable unused ports/services and default accounts to help minimize pathways for malicious attackers.
- Place networked devices behind multiple layers of cyber defenses (such as firewalls, network segmentation, and network intrusion detection and protection).
- Use cyber security best practices (for example: least privilege, separation of duties) to help prevent unauthorized exposure, loss, or modification of data and logs, or interruption of services.

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



# Contents

<b>Safety information</b> .....	<b>1</b>
Important information .....	1
Please note .....	1
<b>Safety precautions</b> .....	<b>1</b>
<b>Contents</b> .....	<b>5</b>
<b>Introduction</b> .....	<b>16</b>
PowerSCADA Expert—A Complete Solution .....	16
Assumptions .....	16
What is SCADA? .....	16
What is PowerSCADA Expert? .....	17
Prepare for PowerSCADA Expert .....	17
Components of a Project .....	17
Configuration Tools .....	17
Runtime Environment .....	18
Supported Device Types and Protocols .....	18
How Do Drivers Work? .....	18
Two Subscription Types .....	18
Subscription Expirations .....	19
Edit Driver Parameters .....	19
Problems with Duplicate Devices using PowerLogic Drivers .....	19
Add a Project Using the Project Setup Tool .....	19
System Definition .....	21
Servers .....	22
When You Set Up More than two I/O Servers per Cluster .....	22
Port Names .....	24
Enter Copy for Translation .....	24
System Requirements and Installation Instructions .....	25
Uninstall and Reinstall PowerSCADA Expert .....	25
IEC 61850 System Setup Workflow .....	25
Get More Information .....	26
Manuals .....	26
Help files .....	26
Customer support .....	26
<b>Use the Migration Utility</b> .....	<b>27</b>
<b>The Profile Editor</b> .....	<b>30</b>
Overview of the Profile Editor .....	31
Typical Workflow Illustration .....	31
Workflow Overview .....	32
Create/Edit Device Type .....	33
Create/Edit Device Profile .....	34
Create/Edit Unit Templates .....	35
Launch the Profile Editor .....	35
Locked and Custom Icons .....	35
Set the Screen Resolution .....	36
Profile Editor Main Menu Options .....	36
Add Engineering Unit Templates, Units, and Conversions .....	37
Set Up Engineering Templates and Select Conversions .....	37
Apply Conversions .....	39
Delete a Template .....	39
Add or Edit a Base Engineering Unit or Conversion .....	40
Edit a Base Engineering Unit or Conversion .....	41
Delete a Base Engineering Unit or Conversion .....	41

Define Device Types and Tags .....	41
Device Type Screens and Workflow .....	41
Use the Define Device Type Tags Tab .....	42
Add, Edit, or Delete a Device Type .....	44
Edit a Device Type .....	45
Delete a Device Type .....	46
Assign Tags to Generic I/O Points .....	46
Print the .CSV File .....	47
Set Up Device Type Categories .....	47
Add a Category or Subcategory .....	48
Edit a Category/Subcategory Name .....	48
Delete a Category or Subcategory .....	48
Edit Functional Addresses .....	49
Add Custom Tags and Tag Addresses .....	49
Set Up Custom Tags .....	49
Edit a Custom Tag .....	51
Delete a Custom Tag .....	52
Edit Tag Addresses .....	52
Real-Time Tag Addresses .....	52
Onboard Alarm Tag Addresses .....	53
Reset Tag Addresses .....	54
Control Tag Addresses .....	55
Edit Address Information .....	56
Add a New Tag Address .....	56
Edit Generic Tag Addresses .....	56
Create Device Profiles .....	57
Enable Waveforms .....	57
View Device Profiles .....	57
Add Edit or Delete Device Profile .....	59
Add a Profile .....	59
Edit a Profile .....	60
Delete a Profile .....	60
Select Trend Intervals .....	61
Set Up Trend Intervals .....	61
To add a trend interval: .....	61
To edit a trend interval: .....	61
To delete a trend interval: .....	61
Edit IEC 61850 Datasets .....	62
Create a New DataSet .....	62
Create a DataSet from an Existing DataSet .....	62
Copy a DataSet to a Device Type .....	62
Edit and Delete DataSets .....	63
Edit IEC 61850 Report Control Blocks .....	63
Create a New Report Control Block .....	63
Create a Report Control Block from an Existing Report Control Block .....	63
Copy a Report Control Block to a Device Type .....	64
Edit and Delete Report Control Blocks .....	64
Create Third Party Modbus Device Type .....	64
Create a Composite Device Type .....	65
Create Data Concentrator Device .....	66
G3200 Device Setup .....	67
For use with multiple devices .....	67
In the Profile Editor .....	67
In CET850 .....	67
In PowerSCADA Expert .....	67
For use with a single device .....	68
In the Profile Editor .....	68

In CET850 .....	68
In PowerSCADA Expert .....	68
DNP3 Protocol Support .....	68
Set Up Projects in the Profile Editor .....	68
Project Screens and Workflow .....	69
Typical Workflow .....	69
The Set Up Projects Tab .....	70
Add, Edit, or Delete a Project .....	70
Adding a Project .....	70
Edit a Project .....	71
Delete a Project .....	72
Edit and Delete Information in a Project .....	72
Customize Tag Names .....	72
Add Project Parameters .....	72
Import and Export Project Files .....	72
Export a Project .....	73
Before you export .....	73
To export a Profile Editor project to the PowerSCADA Expert project: .....	73
To Move Files if the Profile Editor is not with the Server (Reuse of a Project) .....	74
Profile Editor Export .....	74
SCL Export .....	75
Exporting the File .....	75
Import Files into the Profile Editor .....	76
Import SCL Files .....	77
Import Filter Screen .....	77
Functional Constraints .....	77
Report Control Blocks .....	77
Import Reconciliation Screen .....	78
Left-hand pane: .....	79
Middle pane: .....	80
Right-hand pane: .....	80
Complete the Import .....	81
Using Import Templates .....	81
Creating a New Template During Import .....	82
Applying a Template During Import .....	82
Deleting a Template .....	82
Tag Types .....	83
IEC 61850 Tag Construction .....	83
Define an Enumeration .....	84
Use Special Tags to Control Circuit Breaker Status .....	85
Format Code Definitions .....	85
Real-Time Format Code Definitions .....	85
Alarm Format Code Definitions .....	86
Control Format Code Definitions Rules of Operation .....	86
Predefined Control Format Codes .....	87
Predefined Reset Format Codes .....	87
Custom Control and Reset Format Codes .....	87
Logic Codes .....	87
Block Writes .....	98

<b>Add a Project Using the Project Setup Tool</b> .....	<b>99</b>
System Definition .....	100
Servers .....	101
Users .....	102
Menus and Display Pages .....	103
Device Profiles .....	104
Summary .....	105
Devices .....	105
Finish Page .....	106
Parameters Changed In Project Setup .....	106
<b>The Design Time Environment</b> .....	<b>108</b>
Typical Workflow .....	108
The PowerSCADA Expert Pages .....	109
Create a Project .....	109
Reuse Projects Created in the Profile Editor .....	109
Before You Begin .....	110
Add a New Graphics Page .....	110
Change the Background Color of Pages .....	110
Change the Genie Color in Project Pages .....	111
Set a New Page as the Project Startup Page .....	111
Create a One-Line on a Graphics Page .....	111
Enable Lockout/Tagout .....	113
Use Menu Configuration to Edit Pagemenu.dbf (Change the Graphics Page Appearance) .....	114
Create New Genies .....	115
To create a copy of a genie: .....	115
To create a custom symbol for a custom genie: .....	116
Set Up IEC 61850 Advanced Control .....	116
Enable the Advanced Control .....	116
Delete Information from PowerSCADA Expert .....	117
Manage I/O Devices in the Project .....	117
Before You Use the I/O Device Manager .....	117
Define One I/O Device in a Project .....	118
Add a TCP Device to a Project .....	118
Add the TCP Device .....	118
Add a Serial Device to a Project .....	120
Adding the Serial Device .....	121
Add a DNP3_TCP Device .....	123
Add the DNP3_TCP Device .....	123
Add an IEC 61850 Device to a Project .....	125
Add the IEC 61850 Device .....	126
LDName .....	127
BRCBs and URCBs .....	127
Remove a Single I/O Device From a Project .....	127
Use a CSV File to Define Multiple Devices .....	128
Create a CSV File to Add Devices .....	129
Port Names .....	131
Add a Comment Line .....	131
Use a CSV File to Define Multiple Devices .....	132
Update One or All Devices in Project .....	132
CSV File Samples .....	133
Use PowerSCADA Expert One-Lines .....	134
One-Line Flowchart .....	134
What Are PowerSCADA Expert One-Lines? .....	135
Where are the files located? .....	136
One-Line Device (zOL) .....	136
One-Line Colors .....	137
Assigning Colors .....	137

Add INI Settings to AdvOneLine.ini and Citect.ini .....	137
AdvOneLine.ini Settings .....	137
Citect.ini Settings .....	139
Start and Stop One-Lines .....	139
Work with Genies .....	139
Genie Type Descriptions .....	139
Configure an Automatic Transfer Switch (ATS) .....	140
Transfer Switch Information: Left/Right/Bottom .....	140
Display Information .....	140
Configure a Busbar .....	140
Configure a Circuit Breaker or Switch .....	141
Configure a Meter .....	141
Configure a Source .....	142
Configure a Transformer .....	142
GenieDefinitions.xml File .....	143
SupportedGenies.xml File .....	143
Work in the Configuration Utility .....	143
One-Line Configuration Utility .....	144
Modify AdvOneLine.csv .....	144
Review Genie Configurations .....	144
One-Line Errors and Warnings .....	145
Communication Errors .....	146
Error Logging .....	146
Perform Repairs for One-Lines .....	147
Automatic Transfer Switch (ATS) Information .....	148
ATS Information .....	148
ATS Errors .....	149
Busbar Information .....	149
Busbar Errors .....	149
Breaker and Switch Information .....	150
Breaker and Switch Errors .....	150
Meter Information .....	151
Meter Errors and Warnings .....	151
Source Information .....	151
Source Errors and Warnings .....	152
Transformer Information .....	152
Transformer Errors .....	153
Add Custom Images to Graphics Pages .....	153
Work with Alarms .....	154
Alarms Overview .....	154
PC-Based Alarms .....	154
Onboard Alarms .....	155
Set Up Alarm Pages .....	155
Add Setpoints and Delays .....	155
Set Up an Alarm Based on an Enumeration .....	155
Change an Alarm Severity .....	156
Enable Waveforms for Onboard Alarms .....	156
Set Parameters for Event Log Length and Historical Logging of Events .....	156
Event Storage: [Alarm]SummaryLength Parameter .....	156
Add an Onboard Alarm Tag .....	157
Set Up Audible Alarms .....	157
Set Up Tag Viewer .....	158
Restore and Back Up Projects .....	158
Restore a Project .....	158
PowerSCADA Expert Backup .....	158

Use Cicode to Customize a Project .....	159
Use Cicode in PLS_Include .....	159
PLSProviderEngine.ci Module .....	159
Module Construction .....	159
CallProvider .....	159
GetProviderStatus .....	160
GetProviderResult .....	160
Clear Cache and Refresh Platform .....	161
PLS_ClearCache .....	161
PLS_PlatformRefresh .....	162
Customize Default Behaviours .....	162
Time Zone Settings .....	162
Time Synchronization .....	163
Trend Tag Scan Intervals .....	163
Disk Storage Calculation for Trends .....	164
Deadbands and Ignored Devices and Topics .....	164
<b>Waveform Management .....</b>	<b>166</b>
Waveform Storage .....	166
Waveform Database and Special Waveform Tags .....	166
<b>The Runtime Environment .....</b>	<b>168</b>
Open Firewall Ports for PowerSCADA Expert Runtime .....	169
Launch and Close the Runtime Environment .....	169
View the Graphics Page in the Runtime Environment .....	170
Log On to the Runtime Environment .....	170
View the Interface .....	170
Lockout/Tagout Icon .....	171
Communications Loss .....	171
View the Alarms/Events Page .....	172
Equipment Column .....	172
Filter Information .....	173
Remove/Insert/Move Columns .....	173
Sort by Column .....	173
The Event Log .....	173
The Alarm Log .....	173
Unacknowledged Alarms and Disabled Alarms .....	174
Acknowledge, Silence and Print .....	174
When Alarms do not Display Correctly .....	174
Event/Alarm Log Columns Table .....	175
Alarm/Event Filter Form .....	176
Use Security Viewer .....	178
Security Viewer Filter .....	178
Use the Analysis Page .....	179
Use the Equipment Pop-Up Page .....	180
Perform IEC61850 Advanced Control .....	182
View Waveforms .....	182
Enter Setpoints for Alarms .....	182
View Real-Time Trends .....	182
View Lists of Real-Time Information for the Genie .....	183
Override Tag Status .....	183
View the Tag Viewer .....	184
Perform IEC 61850 Advanced Control .....	185
EcoStruxure WebServices (EWS) .....	186
Use Basic Reports .....	187
Set up the Runtime Environment for Basic Reports .....	187
Create the Menu Items for Report Page .....	187
Add the Following INI Parameters .....	188
Set Up a Display Client for Basic Report Viewing .....	188

Tasks in the Basic Reporting Application .....	188
Create and View Basic Reports .....	189
Run a New Basic Report .....	189
Run a Basic Report and Save its Configuration .....	189
View a Basic Report Using a Saved Configuration .....	190
Modify and View a Basic Report Using a Saved Configuration .....	190
Remove a Saved Configuration .....	191
Configure Email Settings to Send Basic Reports .....	191
Send Basic Reports via Email .....	192
Scheduling Basic Reports .....	194
Read, Export, Print, and Edit Basic Reports .....	194
Toolbar Options .....	194
Export a Basic Report .....	195
Edit the Basic Report Appearance .....	195
PowerSCADA Expert Basic Reports .....	195
Single Device Usage Reports .....	196
Multi Device Usage Reports .....	197
Tabular Reports .....	197
Trend Reports .....	198
Enable Windows Authentication for Basic Reporting .....	198
URL Routing for Basic Reports .....	199
Windows 2008 R2 and Windows 7 .....	199
Windows XP .....	200
Localization for Basic Reports .....	200
Template Editor .....	200
Use LiveView .....	200
LiveView Tables .....	201
LiveView Basic Readings Summary .....	201
LiveView Power Flow Summary .....	202
LiveView Energy Summary .....	202
LiveView Energy Readings .....	202
LiveView Fundamental Phasor Readings .....	203
LiveView THD Current Summary .....	203
LiveView THD Voltage Summary .....	203
LiveView Uptime Summary .....	204
LiveView Incremental Reactive Energy Summary .....	204
LiveView Incremental Real Energy Summary .....	204
LiveView Harmonic Apparent Power Flows .....	204
LiveView Harmonic Reactive Power Flows .....	205
LiveView Harmonic Real Power Flows .....	206
LiveView Demand Current Summary .....	206
Live View Demand Voltage Summary .....	206
LiveView Viewer .....	207
Open LiveView from a URL Link .....	207
LiveView Viewer Display .....	207
Where's My Device? .....	209
Missing Topics .....	209
Clear Cache and Platform Refresh .....	209
Set Up LiveView .....	209
Create Menu Item for LiveView Page .....	210
Create a LiveView Template .....	211
LiveView Formatting .....	211
LiveView Placeholders .....	212
LiveView Formulas .....	213
LiveView Thresholds .....	214
Modify LiveView Template .....	215
Duplicate LiveView Template .....	215

LiveView Delete .....	216
LiveView Template Localization .....	216
Enable Windows Authentication for LiveView .....	216
Rapid Access Labels .....	217
Before You Begin .....	217
Create the Sticker .....	218
Read the Sticker .....	218
Troubleshooting .....	219
Multi-Monitor Support .....	219
The Demo Mode .....	219
Application Configuration Utility .....	220
Application Services .....	221
Diagnostics .....	221
EcoStruxure Web Services Setup .....	222
Avoiding EWS Provider Timeouts .....	222
Event Notification Service .....	222
Application Services Host—Citect Data Platform .....	223
Applications .....	223
One-Line Engine .....	224
Basic Reports .....	225
Diagnostics Page/I/O Device Settings .....	225
Security .....	226
One-Time Password (Two-Factor Authentication) .....	226
Add the Citect Parameter .....	228
Autoconfigure the YubiKey .....	228
Program the YubiKey .....	230
Import and Export One-Time Password Settings .....	233
Disabling YubiKeys .....	234
Log in with a Programmed YubiKey and One-Time Password .....	234
Single Sign On .....	235
Single Sign-On (SSO) Calls from a Web Client .....	235
<b>Assign and Control User Privileges .....</b>	<b>236</b>
Default User Access Settings (Privileges) .....	236
Add and Modify User Accounts .....	237
Use Windows Integrated Users .....	237
Change Access Rights .....	238
<b>Appendix 1: Default Genie Library .....</b>	<b>239</b>
PLS_ALARM .....	239
PLS_ANSI_BASE_1 / PLS_ANSI_BASE_2 .....	239
PLS_ANSI_CB_1 / PLS_ANSI_CB_2 .....	241
PLS_ANSI_SW_1 / PLS_ANSI_SW_2 .....	242
PLS_DISPLAY .....	242
PLS_GEN_BASE_1 / PLS_GEN_BASE_2 .....	243
PLS_GEN_CMD_1 / PLS_GEN_CMD_2 .....	244
PLS_GEN_EQ_1 / PLS_GEN_EQ_2 .....	244
PLS_IEC_BASE_1 / PLS_IEC_BASE_2 .....	245
PLS_IEC_CB_1 / PLS_IEC_CB_2 .....	245
PLS_IEC_SW_1 / PLS_IEC_SW_2 .....	246
PLS_METER .....	246
ITEM1 .....	247
<b>Appendix 2: Citect INI Parameters .....</b>	<b>248</b>
Parameters Database .....	248
General PowerSCADA Expert Parameters .....	249
watchtime .....	249
kernelStatisticUpdateRate .....	249
UseWriteMultiRegistersOnly .....	249
timeout .....	250



retry	250
RetryTimeout	250
RetryException	250
standbyRefreshRate	251
standbyCheckTime	251
statusUnitCheckTime	251
initUnitCheckTime	252
initCacheTimeout	252
cacheRefreshTime	252
TimeSync	253
StatusRegister	253
StatusRegistersCount	253
StatusRegisterType	254
ModbusBase	254
RegMode	254
timeZone	255
Alarm Parameters	255
UsePLSFilter	255
Security Parameters	256
EnterPasswordForControl	256
Performance Tuning Parameters	256
Bandwidth Allocation Parameters	256
BandwidthAllocation	257
Packet Blocking Optimisation Parameters	257
enableScatteredReads	257
percentBlockFill	257
maxBlockSize	258
minBlockSize	258
Tag Scan Rate Parameters	259
HighScanRate	259
LowScanRate	259
Advanced Tag Block Capabilities (Invalid Memory Access Blocks defined)	259
Invalid Block Tag Definition Syntax	260
Configuration Notes	261
Write-only Memory	261
Tag Blocking Notes	262
Write-only Tags	263
Waveform Parameters	263
WaveformsDisable	264
WaveformMatchMargin	264
WaveformCheckTime (PM, CM, and Sepam)	264
WaveformZone (Sepam)	264
Sepam Event Reading Parameters	265
EventTable	265
EventIdle	265
Sepam Device Driver INI Configuration Settings	265
MicroLogic Modules Configuration Parameters	266
IFE/IFM	266
MicrologicType	266
CCM	266
Module-Specific Packet Blocking Optimisation Settings	266
MicrologicV INI Settings	267
Data Replication Parameters	267
Database root folder path	267
Database root UNC path	267
Replication destination configuration	268

Graphics Library Parameters .....	268
Maximum number of entries that can be held in Event Log .....	268
Parameters for Alarm and Event States .....	268
Integration and Single Sign-On Parameters .....	269
<b>Appendix 3: Additional INI Parameters .....</b>	<b>271</b>
Quantum PLC Time-Stamped Events .....	271
<b>Appendix 4: Glossary .....</b>	<b>272</b>
Appendix 5: Frequently Asked Questions (FAQs) .....	279
If I don't use PowerLogic drivers, how do I create device profiles? .....	280
How should we manage categories and subcategories? .....	280
When should I create a device type rather than device profile? .....	280
How do we synchronize a new PC with the master Profile Editor PC? .....	280
What do I do before I add or remove devices in the I/O Device Manager? .....	280
What are the requirements for device names? .....	281
How do I troubleshoot device communications issues? .....	281
How do I use MODBUS communications methods? .....	281
What are the naming conventions for servers and clusters? .....	282
How and when do I create users for the Runtime environment? .....	282
How do I manage projects in the PowerSCADA Studio? .....	282
On the Graphics page, what do I need to know about creating genies? .....	283
How do we customize existing templates? .....	284
How do I change the default pickup/dropout text for alarms? .....	285
What can I modify during runtime? .....	285
Why do the browser navigation buttons not work? .....	285
What can I set up in logging and archiving? .....	285
How do I create and configure busbars? .....	285
What INI parameters should I use for debugging? .....	286
How do I tune my system for best performance? .....	287
If a tag is configured, how is it polled in the device? .....	288
Device popup from a one-line: Why do the fields overlap? .....	289
Can I change the %CLUSTER% name in the I/O Device Manager? .....	289
A device can prevent writes to its registers: how do I ensure that writes are successful? .....	289
How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes? .....	290
How do I create an audit in the Event Log for user logins and logouts? .....	290
Why am I seeing #COM for circuit breaker status in the genie status page? .....	290
Why can't I acquire waveforms in the waveform viewer? .....	291
Why won't the Excel DBF Add-In toolbar install? .....	291
What causes the "First dbf record" error message? How do I keep it from happening? .....	291
Why is my device in comms loss? .....	291
How do I set up select before operate? .....	291

<b>Appendix 5: Frequently Asked Questions (FAQs)</b> .....	<b>293</b>
If I don't use PowerLogic drivers, how do I create device profiles? .....	293
How should we manage categories and subcategories? .....	293
When should I create a device type rather than device profile? .....	293
How do we synchronize a new PC with the master Profile Editor PC? .....	293
What do I do before I add or remove devices in the I/O Device Manager? .....	293
What are the requirements for device names? .....	294
How do I troubleshoot device communications issues? .....	294
How do I use MODBUS communications methods? .....	295
What are the naming conventions for servers and clusters? .....	295
How and when do I create users for the Runtime environment? .....	295
How do I manage projects in the PowerSCADA Studio? .....	295
On the Graphics page, what do I need to know about creating genies? .....	296
How do we customize existing templates? .....	297
How do I change the default pickup/dropout text for alarms? .....	298
What can I modify during runtime? .....	298
Why do the browser navigation buttons not work? .....	298
What can I set up in logging and archiving? .....	298
How do I create and configure busbars? .....	298
What INI parameters should I use for debugging? .....	299
How do I tune my system for best performance? .....	300
If a tag is configured, how is it polled in the device? .....	301
Device popup from a one-line: Why do the fields overlap? .....	302
Can I change the %CLUSTER% name in the I/O Device Manager? .....	302
A device can prevent writes to its registers: how do I ensure that writes are successful? .....	302
How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes? .....	303
How do I create an audit in the Event Log for user logins and logouts? .....	303
Why am I seeing #COM for circuit breaker status in the genie status page? .....	303
Why can't I acquire waveforms in the waveform viewer? .....	304
Why won't the Excel DBF Add-In toolbar install? .....	304
What causes the "First dbf record" error message? How do I keep it from happening? .....	304
Why is my device in comms loss? .....	304
How do I set up select before operate? .....	304
<b>Index</b> .....	<b>306</b>

## Introduction

Welcome to the help file for PowerSCADA Expert, v8.2. 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 16

[Assumptions](#) on page 16

[What is SCADA?](#) on page 16

[What is PowerSCADA Expert?](#) on page 17

[Prepare for PowerSCADA Expert](#) on page 17

[Components of a Project](#) on page 17

[Configuration Tools](#) on page 17

[Runtime Environment](#) on page 18

[Supported Device Types and Protocols](#) on page 18

[System Requirements and Installation Instructions](#) on page 25

[Uninstall and Reinstall PowerSCADA Expert](#) on page 25

[IEC 61850 System Setup Workflow](#) on page 25

[Get More Information](#) on page 26

[Manuals](#) on page 26

[Help files](#) on page 26

[Customer support](#) on page 26

## PowerSCADA Expert—A Complete Solution

The PowerSCADA Expert system provides a complete solution for the monitoring and control of any electrical distribution network. Using PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards 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 16

[What is SCADA?](#) on page 16

[What is PowerSCADA Expert?](#) on page 17

## 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 I/O Device Manager, 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 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 Graphics Builder (runtime mode), users can view the final one-line drawings, including alarms, events, and history data. With the appropriate degree of password-controlled authority, users can also perform advanced tasks, such as changing alarm setpoints and racking devices in and out.

## Prepare for PowerSCADA Expert

For a full discussion about preparation for a project, see **Plan a Project** in the CitectSCADA.chm help file (Program Files > Schneider Electric > PowerSCADA Expert > v8.2 > bin). This section of the help discusses physical layout, requirements such as architecture and security, and project design.

## Components of a Project

For detailed information about the components that make up a project, see **Components of a project** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin). The project components include graphic components, tags, alarms, system components, communications components, I/O Server components and Cicode/CitectVBA.

## Configuration Tools

Configuration tools consist of:

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

**I/O Device Manager:** Using this tool, 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.

**PowerSCADA Studio:** Use the Studio for basic navigation. From here, you also choose the active project. Use it also 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 Graphics Builder to create one-line drawings that users can view in the runtime environment. These drawings are populated with interactive objects that are generated by genies. You can also use the graphics tool to set up system alarms and trends.

## Runtime Environment

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

## Supported Device Types and Protocols

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

- Generic MODBUS (includes BCPM and any device, such as a PLC or UPS, that communicates via MODBUS). When adding a controllable device in the Profile Editor, such as a circuit breaker, use the “Controllable Device” driver; otherwise, use the “Generic Power Device” driver. For JBus devices, select Generic JBus Device.
- Sepam 20, 40, and 80 Range, 2000
- Masterpact MicroLogic 5P and 6P, A, H
- Compact NSX (MicrologicV)
- Micrologic MTZ (MicrologicX)
- CM2000
- CM4000 series
- PM650
- PM800 series
- PM5000 series
- PM700 series
- ION protocol devices
- IEC 61850 protocol devices
- IEC 870-5-104
- DNP3
- BCPMA (branch circuit power meter, full feature support)
- CSI SER (Cyber Sciences SER)
- ProTime 100 SER (Monaghan Engineering)

## How Do Drivers Work?

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

### Two Subscription Types

There are two different types, one used between the graphics level and I/O server, and one for polling devices and cache refreshing. The subscription between drivers and polling devices does not increase point count. Only the subscription that begins at a client system and ends up in the I/O server will increase point count. Via this subscription, requests are sent to the drivers with value changes propagating all the way back to the client system. The client system could be the display client, alarm server, trend server and so on. What a driver then chooses to do with the requests—in

terms of coupling this to a physical request to a field device—can differ, depending on the protocol. Some simple protocols propagate the request straight through to the field device; others have their own polling scheme to the field device and merely service the driver requests from a cache.

### Subscription Expirations

If a tag is no longer being read, the cache refreshes in this manner: Graphics client subscriptions are immediately unsubscribed when the graphics page is closed. Although most drivers release subscriptions if no client is requesting them, the IO Server 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 IO Server. On the other hand, once a subscription goes against the point count, it remains in the count as long as the project is running.

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

## Edit Driver Parameters

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

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

To begin editing datasets, click Edit in the DataSets line. Follow instructions in [Edit IEC 61850 Datasets on page 62](#) 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 63](#) 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.

## Add a Project Using the Project Setup Tool

There are two methods for adding a project:

You can use the PowerSCADA Studio, (see *The Design Time Environment*), which is a more manual process. Use PowerSCADA Studio to ensure that you have covered everything, and for further modification of your project.

Or, you use this tool to rapidly deploy a project. Using Project Setup, you can do the following:

- create and name a project
- select screen resolution and determine whether you will use normal or high contrast
- specify primary and secondary server connections
- specify the Advanced Reports and Dashboards connection
- add users and Windows authentication
- add devices
- add default pages to your project
- add menus to display pages in runtime
- choose the landing page for each monitor in a multi-monitor project

If you are using optional components (see below), you need to install them separately.

After you create the project and its features, you can edit other settings, such as devices in the CSV file, and use it to update your project.

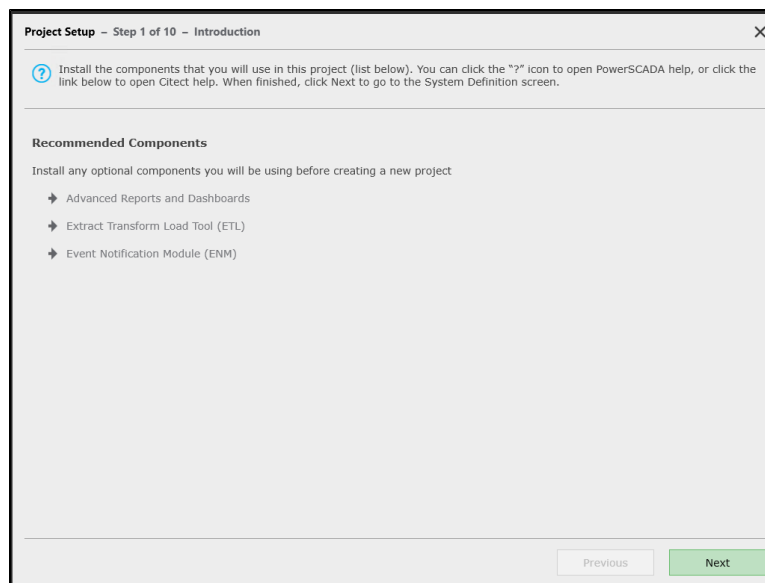
You save time when you use Project Setup, as you are able to change a variety of parameters. When you do not use Project Setup, you then must navigate to a variety of locations to change or set up these parameters. For a list of parameters that you can set in Project Setup, see:

[Parameters Changed In Project Setup on page 106](#)

For help on any of the Project Setup pages, you can:

- click "?" to the left of the top line to view the entire PowerSCADA Expert help file
- hover your mouse over fields to read tooltips

To launch Project Setup, launch the PowerSCADA Studio. From the Projects tab, click Add > Create and Configure Project. You see the Introduction page.





This page lists optional components that you may want to install. The install files are on the root of the PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards installation media.

- PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. Use this component to view advanced reports and dashboards from Power Monitoring Expert. Install this component from the PowerSCADA Expert installation media.
- Extract Transform Load (ETL) module. Use this component to extract reporting information from PowerSCADA Expert and transfer it to Power Monitoring Expert, for use in reports).
- Event Notification Module (ENM). Use ENM to set up email and text messaging for project alarms.

Click Next to display the System Definition screen.

## System Definition

The screenshot shows a software window titled "Project Setup - Step 2 of 10 - System Definition". At the top, there is a help icon and a text box: "Name your project or choose an existing project to edit. Provide screen resolution and background appearance. After you click Next, the project is created; and it can only be changed or deleted in the Citect IDE." Below this, the "PowerSCADA Project" section has two radio buttons: "Create New" (selected) and "Edit Existing". A text input field contains "Project1". The "Resolution (Aspect Ratio)" section lists five options: "1024 X 768 (4:3)", "1280 X 1024 (5:4)", "1400 X 900 (8:5)", "1680 X 1050 (8:5)", "1920 X 1080 (16:9)" (highlighted in green), and "1920 X 1200 (16:10)". The "Style" section lists two options: "Standard" (highlighted in green) and "High Contrast". At the bottom right, there are "Previous" and "Next" buttons.

On this screen, you determine whether you are creating a new project or editing an existing one, and the "view" of the project.

1. Click either Create New or Edit Existing.
2. If you click Create New, type a project name. If you click Edit Existing, choose a project from the list that displays. Use only alphanumeric characters and underscores.
3. In the Resolution box, choose the screen resolution that you want for the graphics pages in this project. This should match the resolution of the monitor that will display graphics pages.
4. In the Style box, choose whether you want Standard or High Contrast. Standard uses a white background. High contrast uses a black background, which makes it easier to view graphics pages.
5. Click Next to display the Servers page.

**NOTE:** After you click Next, the project is created. You cannot change or delete it here. To make changes or to delete it, you must use the PowerSCADA Studio.

You can also set high contrast via the parameters in the PowerSCADA Studio. Open your project in the PowerSCADA Studio, then click Settings > Parameters. The parameter name is **IsHighContrast**. 0 = normal; 1 = high contrast.

## Servers

**Project Setup – Step 3 of 10 – Servers and Web Client**

Enter the address information for the servers in this project. If two or more servers are in the startup project file, you can only add an address and user account for Advanced Reports and Dashboards.

**Topology**

Server Name or IP Address  
127.0.0.1

Redundant System  
Standby Server Name or IP Address

Advanced Reports and Dashboards  
Advanced Reports Server Name or IP Address

User Name Password Confirm Password

Previous Next

Use this screen to define the server information for your primary server and the server for Advanced Reports and Dashboards.

The Project Setup detects the number of servers that are in your starter project. If you only have one server (for example, using the loopback IP address), you see all of the fields in the screen above. If you are using a project that has two or more servers identified, you only see the bottom section, Advanced Reports and Dashboards.

1. Server Name or IP Address: Type the Server Name or IP Address for the project's primary server, or select it from the drop-down list.
2. Redundant System: If this is a redundant system, check Redundant System. Checking this box enables the Standby Server Name or IP Address prompt.
3. Standby Server Address: Type the Server Name or IP Address of the standby server, or select it from the drop-down list.
4. Advanced Reports and Dashboards: If you have installed the Advanced Reports and Dashboards component (WebReach is also assumed to be on this server), check this box, which will enable the Advanced Reports prompts.
5. Advanced Reports Server Name or IP Address: Type the Server Name or IP Address of the Advanced Reports server, or select it from the drop-down list.
6. User Name/Password/Confirm Password: These fields are only enabled after you check Advanced Reports and Dashboards. Type the user name and password used for the Advanced Reports and Dashboards server. (Note that you must enter a user name and password if you check the Advanced Reports and Dashboards box.)  
Retype the password in the Confirm Password field.
7. Click Next to view the Users 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 I/O Device Manager configuration (see [Add the Cluster, Network Address, and Servers](#)). 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 I/O Device Manager
- 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> (example: Px\_BOARDx\_PRJz)

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 I/O Device Manager. 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.

## Port Names

The I/O Device Manager 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 wizard, you can name ports. See [Define One I/O Device in a Project on page 118](#) for more information.

## Enter Copy for Translation

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

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

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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.

## System Requirements and Installation Instructions

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

## Uninstall and Reinstall PowerSCADA Expert

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

- PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards (if you uninstall this, you also uninstall the Profile Editor)
- PowerSCADA Expert Profile Editor
- Any additional PowerSCADA Expert programs, such as the WebServer, that you installed

If you uninstall programs after you have already created projects, the project data will not be deleted. It is in [Project Drive]\Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. The first time you launch the application after you re-install it, it will locate the project data and re-link it.

## 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 67](#)).
2. Import the first ICD file into the Profile Editor (see [Import Filter Screen on page 77](#)).
  - a. Create the device type.
  - b. Match or verify tags for PowerSCADA Expert.
  - c. Complete the import.
3. Create a device profile for the IEC 61850 device type (see [Add an IEC 61850 Device to a Project on page 125](#)).
  - a. If needed, add/edit datasets and report control blocks (see [Edit IEC 61850 Datasets on page 62](#) and [Edit IEC 61850 Report Control Blocks on page 63](#)).
  - 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 I/O Device Manager.

- 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 I/O Device Manager, add your devices to PowerSCADA Expert.

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

For more information, see [Add an IEC 61850 Device to a Project on page 125](#).
- 10. Compile and run the project.

## Get More Information

For additional information, use one of these options:

[Manuals on page 26](#)

[Help files on page 26](#)

[Customer support on page 26](#)

## Manuals

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

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

## Help files

In addition to the help file released with this product, there are several related help files. They are located in the PowerSCADA Expert Bin folder: \Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Bin).

## Customer support

The following is the contact information for customer support:

Schneider Electric

35, rue Joseph Monier

CS 30323

F - 92506 Rueil Malmaison Cedex

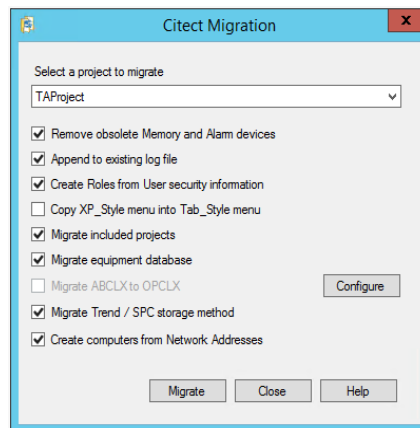
[www.schneider-electric.com](http://www.schneider-electric.com)

## Use the Migration Utility

The migration utility allows you to migrate previous versions of PowerSCADA Expert to version 7.3. You only need to run this utility one time. This is a single migration that updates system data. All future upgrades require that you back up your project, uninstall the old version, install the new version, and restore your database.

The table below describes the changes that will be made.

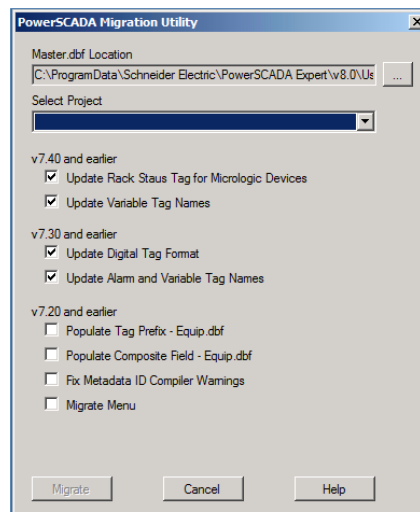
1. Before you run the migration utility, back up your system.
2. If necessary, restore the project that you want to migrate (use the Restore function from PowerSCADA Studio > Projects tab > Backup > Restore).
3. Launch the Citect Migration Tool in PowerSCADA Studio: on the Projects page, click Migration Tool > Citect Migration Tool.
4. From the Citect Migration window, select the options shown below:



5. Click Migrate.

When the migration is complete, an information window displays information about what was converted or upgraded, and where the resulting log file is stored.

6. Launch the PowerSCADA Migration Utility: from the Projects tab > click Migration Tool > PowerSCADA Expert Migration Tool.



7. From the Master.dbf Location drop down list, choose the location for the Master.dbf.
8. From the Select Project drop down list, choose the project that you are migrating.

9. In the bottom section, check the boxes for the elements you want to update in the runtime database (see table below for descriptions).

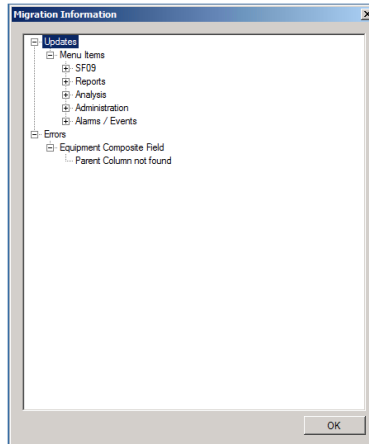
**NOTE:** The items with asterisks will be updated in the Equipment.profiles file at every migration. When you check an item with an asterisk, it will also update the Profile so that future information added to it will be in sync with the current version. So, for example, if you run the migration and check "Update Variable Tag Names," future variable tags will be correctly formatted for the current version.


10. Click Migrate.
11. Verify that you have backed up the project, then click Yes.
12. If there is already a PageMenu.dbf file that is creating a menu for your graphics pages, you see a message telling you that the PageMenu.dbf is not empty. Click Yes to override this file, which overwrites the menu, leaving it blank. Click No to retain the menu for version 7.4.

Element	Description	Changes
<b>v7.30 and earlier:</b>		
Update Digital Tag Format	This option will update all DIGITAL tags in Variable.dbf to FORMAT "##."	This causes digital tags in PowerSCADA Expert to display without decimals ("1" or "0," but not "1.000.")
Update Alarm and Variable Tag Names	Previously, some device-specific tags were renamed to fit a generic naming convention for version 7.40. This option renames all existing tags to the new convention names.	Check this box to rename all new convention names. For example, the old "Sepam Not Reset" is now "Generic Not Reset."
<b>v7.20 and earlier:</b>		
Populate Tag Prefix-Equip.dbf	Equipment Name, which was used to build tag names, is now the equipment hierarchy name (can no longer be used to build tag names)	TagPrefix field added. It is now used to build tags. If the TagPrefix field is empty, IODevice name is used to populate Tag Prefix. If IODevice name is also empty (in a composite device), EquipmentName is used IF there are no periods in the name.
Populate Composite Field - Equip.dbf	The Parent field (previously used to determine the parent piece of equipment) has been removed from the .dbf file.	The Composite field replaces the Parent field. The Composite field will display the Parent field information, if applicable.
Fix Metadata ID Compiler Warnings	The Cicode function StrToLocal no longer allows partially translated text. For example, in @(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).

When migration is complete, a summary screen lists the results of the migration, including updates and errors. The following is an example:





13. Pack the databases and compile the project (from the Projects tab, click Pack. To compile, click  on the System menu, on the left.
14. After you install version 7.3, you need to:
  - a. Back up the project.
  - b. Uninstall (if you are using the same computer to reinstall).
  - c. Install the new version.
  - d. Add the One-Line device (see [One-Line Device \(zOL\)](#) on page 136).
  - e. Run the Advanced One Line tool (see [Work in the Configuration Utility](#) on page 143).
15. If you are going to upgrade to a later version, you need to:
  - a. Back up your project.
  - b. Uninstall version 7.3.
  - c. Install the new version.
  - d. Restore your project.

# 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 31](#)

[Typical Workflow Illustration on page 31](#)

[Launch the Profile Editor on page 35](#)

[Add Engineering Unit Templates, Units, and Conversions on page 37](#)

[Define Device Types and Tags on page 41](#)

[Device Type Screens and Workflow on page 41](#)

[Print the .CSV File on page 47](#)

[Add Custom Tags and Tag Addresses on page 49](#)

[Add, Edit, or Delete a Device Type on page 44](#)

[Edit Tag Addresses on page 52](#)

[Edit Generic Tag Addresses on page 56](#)

[Edit Functional Addresses on page 49](#)

[Create Device Profiles on page 57](#)

[Enable Waveforms on page 57](#)

[View Device Profiles on page 57](#)

[Add Edit or Delete Device Profile on page 59](#)

[Edit IEC 61850 Datasets on page 62](#)

[Edit IEC 61850 Report Control Blocks on page 63](#)

[IEC 61850 System Setup Workflow on page 25](#)

[Create Data Concentrator Device](#)

[G3200 Device Setup on page 67](#)

[Set Up Projects in the Profile Editor on page 68](#)

[Project Screens and Workflow on page 69](#)

[The Set Up Projects Tab on page 70](#)

[Add, Edit, or Delete a Project on page 70](#)

[Create a Composite Device Type](#)

[Edit and Delete Information in a Project on page 72](#)

[Customize Tag Names on page 72](#)

[Import and Export Project Files on page 72](#)

[Tag Types on page 83](#)

[Customize Tag Names on page 72](#)

[IEC 61850 Tag Construction on page 83](#)

[Define an Enumeration on page 84](#)

[Format Code Definitions on page 85](#)

[Logic Codes on page 87](#)

## 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 41](#) 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 57](#) 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 68](#) for complete instructions.

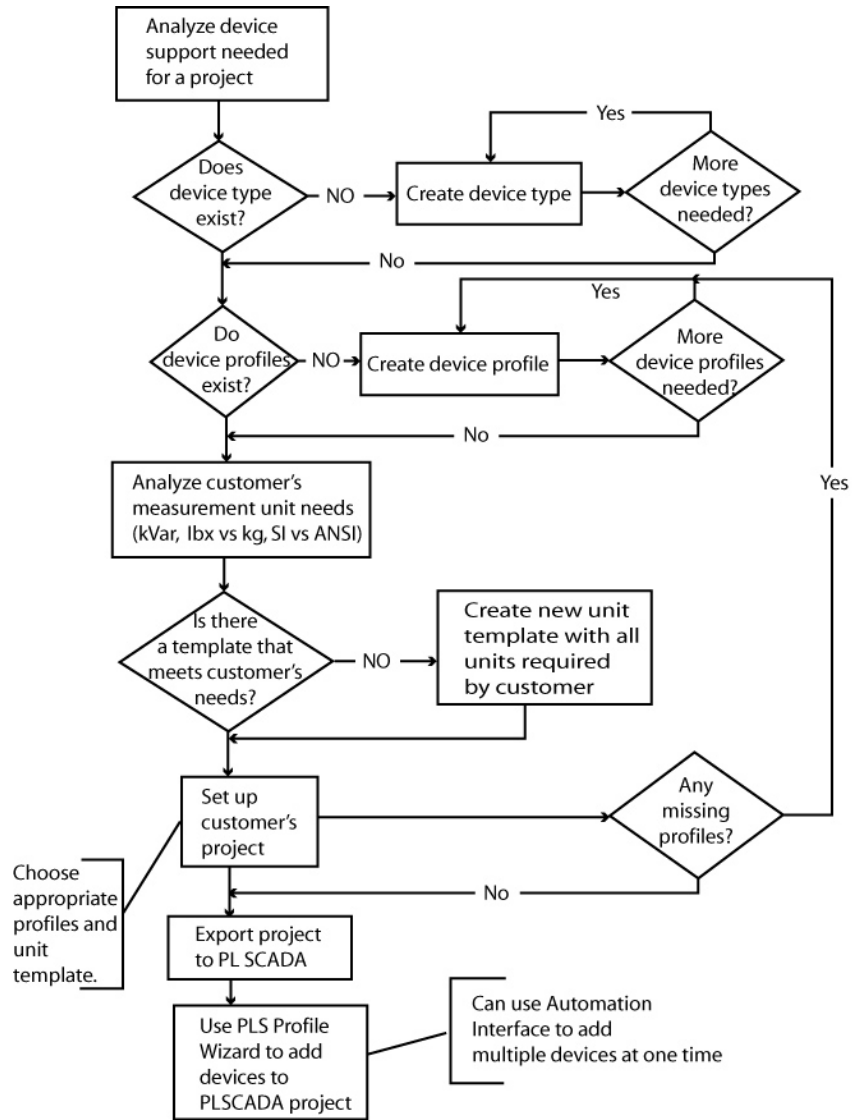
This product uses the IEC61850 tag-naming convention to create tags that measure device quantities. Although most of the tags you will use are already entered into the system, you can add custom tags. For more information, see [Tag Types on page 83](#).

## 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

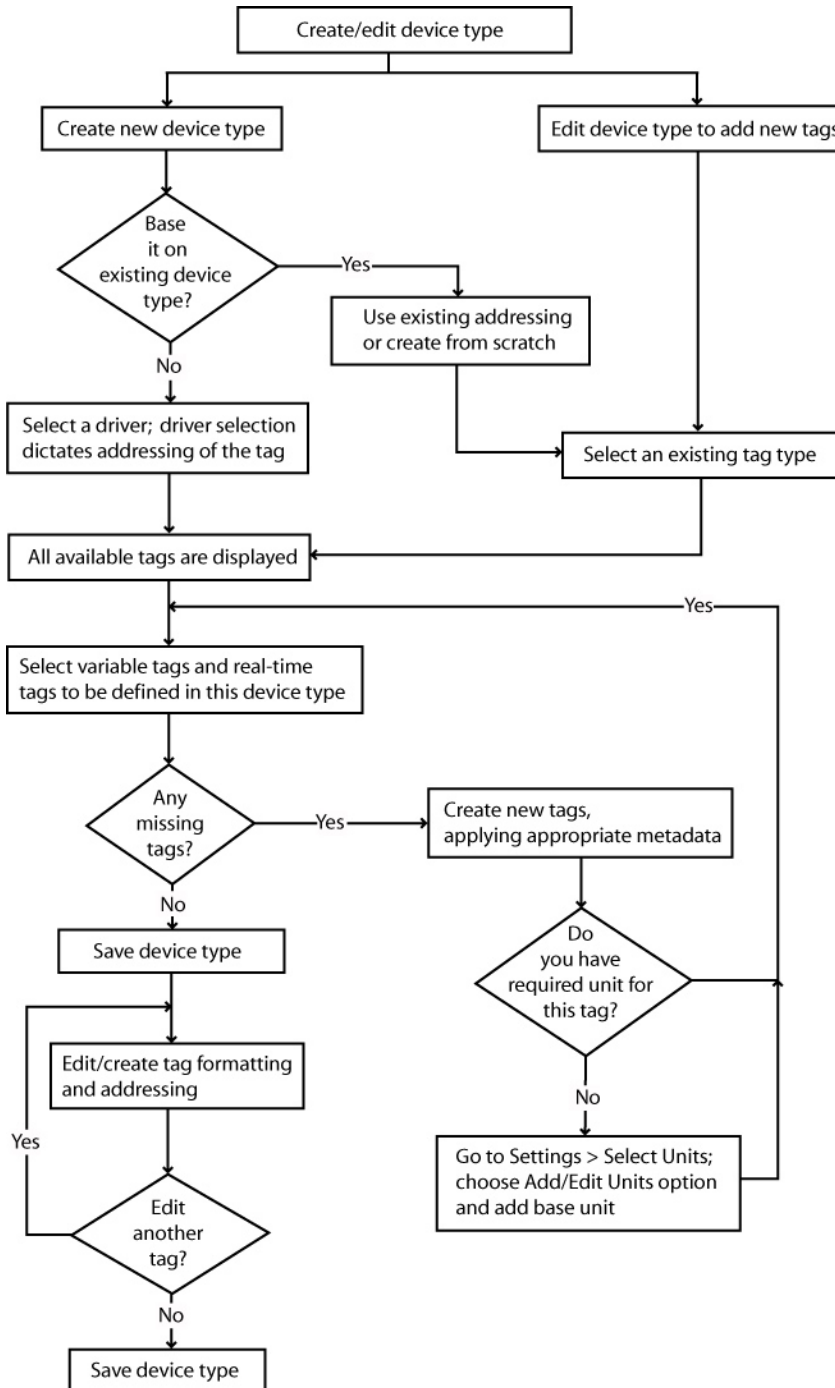
## Workflow Overview



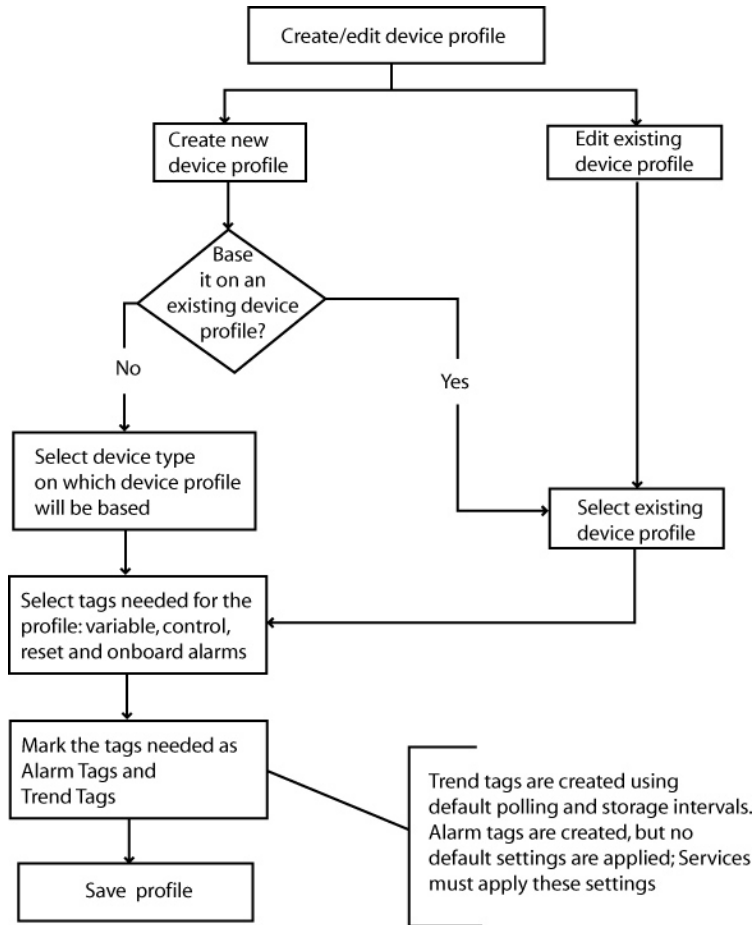
Choose appropriate profiles and unit template.

Can use Automation Interface to add multiple devices at one time

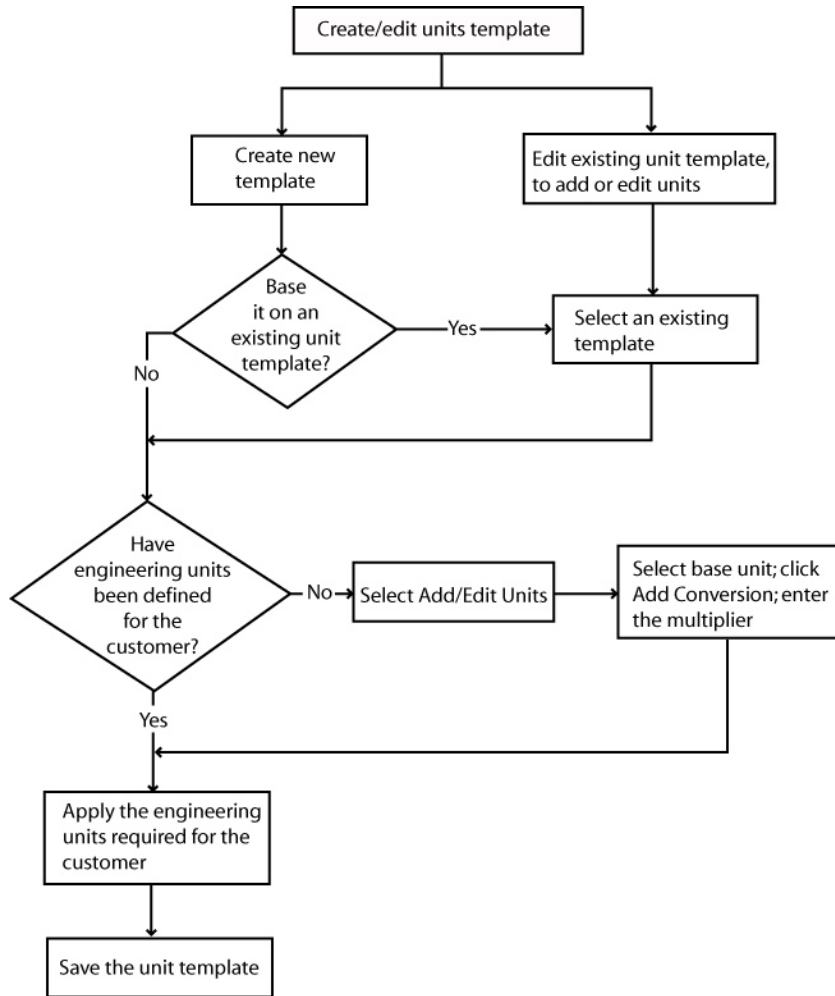
## Create/Edit Device Type



## Create/Edit Device Profile



## Create/Edit Unit Templates



## Launch the Profile Editor

To launch the Profile Editor, click Start > Programs > Schneider Electric > PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards > Config Tools > Profile Editor.

The Profile Editor screen displays with the Define Device Tags tab selected. There are two other tabs, used to create device type profiles and projects.

## Locked and Custom Icons

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

### The Locked Icon :

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

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

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

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

#### The Custom Icon :

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

## Set the Screen Resolution

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

Options are:

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

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

## Profile Editor Main Menu Options

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

Field Name/Valid Entries	Comments
File > Save	Saves any current changes.
File > Create CSV file	Creates .CSV file of basic tag data. Store the file in a folder you designate. Can be viewed in Excel.
File > Print Tag Selections	Displays a print preview of all of the tags for this device type. You can then print the spreadsheet.
File > Import	You can import projects from other instances of the Profile Editor. These must be PLS or ICD files. To begin importing, click File > Import. For ,more information, see <a href="#">Import and Export Project Files on page 72</a> .



Field Name/Valid Entries	Comments
File > Export	You can export a PLS or ICD file to be used in another instance of the Profile Editor, or to be used as a backup. To begin exporting, click File > Import. For more information, see <a href="#">Import and Export Project Files on page 72</a>
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 <a href="#">Using Import Templates on page 81</a> .
Settings > Set Up Custom Tags	Displays the Add/Edit Custom Tags screen. See <a href="#">Set Up Custom Tags on page 49</a> for a description of this screen.
Settings > Set Up Device Type Categories	Displays the Set Up Device Type Categories. See <a href="#">Set Up Device Type Categories on page 47</a> for a description of this screen.
Settings > Set Up Engineering Unit Templates	Displays the Set Up Engineering Unit Templates screen. Click <a href="#">Set Up Engineering Templates and Select Conversions on page 37</a> for a description of this screen.
Settings > Set Up Trend Definitions	Displays the Set Up Trend Definitions screen. Click <a href="#">Set Up Trend Intervals on page 61</a> 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 37](#)

[Add or Edit a Base Engineering Unit or Conversion on page 40](#)

## Set Up Engineering Templates and Select Conversions

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

To view the Set Up Engineering Units screen, click Settings > Set Up Engineering Unit Templates. The following table describes the parts of the Set Up Engineering Unit Templates screen (it assumes that Display 'Advanced' Fields is checked). When you have finished making change, click Save & Exit.

Field Name	Valid Entries	Comments
Template Options box	Create New	Click to begin creating a new engineering unit template.
	Create From	Click to create an engineering units template that is based on an existing template.
	Edit Existing	Only available if you have added a template. Click to edit an engineering unit or its conversion.
	Delete	Only available if you have added a template. Click to begin deleting an engineering unit and its conversion. You cannot delete a locked template.
Unit Template to Create From	From the drop-down menu, select the template you wish to copy, in order to create a new template.	This field is live only when <i>Create From</i> is chosen as the option. The new template will initially include all of the units/conversions of the original; but you can add units and change the conversion settings.
Unit Template Name	This field is blank if you selected <i>Create New</i> or <i>Create From</i> ; type the name of the new template. A name displays if you have selected a template to edit; you can change the name. A name displays, but it is greyed out if you selected a template to delete. Click <i>Save</i> to save the changes you make.	When creating a new template or creating from an existing template, type the name of the new template. To change the name of an existing template, choose it from the <i>Unit Template to Delete</i> menu, then change the name here.
Lock this Template	Click to prevent the template from being edited in the future.	The only way to "edit" a locked template is to delete it, and add back a new one with the edits entered.
Display Associated Projects	Live only when in "Edit" mode. Displays all projects that use this template.	You only need this if you want to delete a template that is associated with a project. Note the projects that display in the list, then go to the Set Up Project tab. For each project that you noted, change the unit template.
Display 'Advanced' Fields	Check this box to display additional columns of information about the template.	Unchecked: displays the unit and its abbreviation only. Checked: displays also the conversion, and its abbreviation and multiplier.
<b>Default Units Subtab</b>		
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 <a href="#">Add or Edit a Base Engineering Unit or Conversion on page 40</a> .
Abbreviation	n/a	Added for the unit when the selected unit was created. To edit a user-created unit, see <a href="#">Add or Edit a Base Engineering Unit or Conversion on page 40</a> .
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 <a href="#">Add or Edit a Base Engineering Unit or Conversion on page 40</a> . Fahrenheit to Celsius temperature conversions must be handled by editing Cicode (Citect.ini).
Abbreviation	n/a	This is abbreviation for the selected unit. When the Selected Unit is changed, this field changes accordingly.

Field Name	Valid Entries	Comments
Multiplier	n/a	Added for the unit and for the conversion when the base unit was created. Pre-defined units/conversions cannot be changed. To edit a user-created unit, see <a href="#">Add or Edit a Base Engineering Unit or Conversion on page 40</a> .
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.
<b>Unit Exceptions Subtab</b>		
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	<ol style="list-style-type: none"> <li>From the dropdown list, choose the unit you want to use for this tag/tag group.</li> <li>Click the radio button for the exception to be made.</li> <li>Either double-click the tag, or click the right arrow to move it to the Exception list.</li> </ol>	<ol style="list-style-type: none"> <li>If you choose Apply Unit Conversion, the tag will be reported according to the unit you select. For example, if you want to report Air Volume in gallons, rather than cubic meters, choose "gallon" from the Select Unit dropdown list.</li> <li>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.</li> </ol>
Exception List	Review your changes.	You can check or uncheck tags here, changing them from one conversion option to the other. When you uncheck a tag, you do not remove it, you change it from being converted to simply being reported according to the unit you selected.

### Apply Conversions

Use this screen to apply unit conversions to a template. To add a new conversion, see [Add or Edit a Base Engineering Unit or Conversion on page 40](#).

To apply a conversion:

- From the main window of the Profile Editor, click Settings > Set Up Engineering Unit Templates.
- Click Edit Existing, then select the template for which you want to select unit conversions.
- 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 40](#)).
- Repeat step 3 for all units that you want to change.
- Click Save to save the change, or click Save & Exit to save changes and close the screen.

### Delete a Template

You cannot delete either the standard template or a locked template.

To delete a template:

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

## Add or Edit a Base Engineering Unit or Conversion

Use the Add/Edit Units screen to add, edit, or delete base units and conversion units for custom tags. You cannot make any changes to predefined units (those that are greyed out).

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

Field Name	Valid Entries	Comments
Unit Conversions	n/a for pre-defined units/conversions (greyed out) Select user-created units to begin edits.	All base engineering units and their conversions display. Greyed-out items are predefined; they cannot be edited or deleted. Note that predefined units can have custom conversions, which are editable.
Add Base	Click to begin adding a new base unit.	The Base Unit and Base Unit Abbreviation fields become live.
Add Conversion	Click to begin adding a conversion to a base unit.	The Base Unit field displays the unit you highlighted; the Conversion Unit, Conversion Abbreviation, and Multiplier fields become live.
Edit Unit/ Delete Unit	Click to either edit a custom unit/conversion, or to delete it.	These buttons are live when you select a custom unlocked unit.
Base Unit	When editing a unit/conversion, select the unit from this drop-down menu. When adding a new base unit, type the name.	Used in the Profile Editor only; not passed to projects for graphics viewing.
Base Unit Abbreviation	Type the abbreviation for the selected base unit.	If there is no conversion, this is passed to projects for viewing graphics.
Conversion Unit	Type the name of the conversion unit, such as milliamps, when amps is the base unit.	Becomes live only when you highlight a unit. Used in the Profile Editor only; not passed to projects for graphics viewing.
Conversion Abbreviation	Type the abbreviation for the conversion unit.	This is passed to projects for viewing graphics.
Multiplier	Use this field to determine the number of base units that are in the conversion unit. Type the multiplier "M," where Conversion Unit x M = Base Unit.	Example: There are 1,000 bytes in a kilobyte; so, the conversion unit multiplier is 1000. If you have 17.3 kB, $17.3 \times 1,000 = 17300$ bytes

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

### Edit a Base Engineering Unit or Conversion

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

To edit a unit or conversion:

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

### Delete a Base Engineering Unit or Conversion

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

To delete a unit or conversion:

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

## Define Device Types and Tags

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

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

In this section, you can learn about:

[Device Type Screens and Workflow on page 41](#)

[Use the Define Device Type Tags Tab on page 42](#)

[Add, Edit, or Delete a Device Type on page 44](#)

[Print the .CSV File on page 47](#)

[Set Up Device Type Categories on page 47](#)

[Edit Functional Addresses on page 49](#)

## 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 40](#).
2. To manage the tags you will use, see [Define Device Types and Tags on page 41](#).

3. To add and edit custom tags, see [Add Custom Tags and Tag Addresses on page 49](#).
4. To add or edit device types, see [Add, Edit, or Delete a Device Type on page 44](#).
5. To establish device type categories and subcategories, used in reporting, see [Print the .CSV File on page 47](#).
6. To edit tag addresses, see [Edit Tag Addresses on page 52](#).

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 83](#).

## Use the Define Device Type Tags Tab

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

Field Name/Valid Entries	Comments
Device Type Name/Select the device type.	Each device type includes a different number of tag categories, which also changes the list of tags that display. The device list includes the default device types, as well as any that have been created for this system.
Add/Edit button/Click to open the Add/Edit Device Type screen.	Provides a means of adding new device types and editing custom device types (user-created device types that are not locked). Also provides a means of adding new custom tags and editing existing tags.
Locked/Custom icons:  	Locked icon indicates that the list of selected tags cannot be edited. Custom icon indicates that the device type was created by a user. See <a href="#">Locked and Custom Icons on page 35</a> for complete information.
<b>Tag groups (left-hand pane)</b>	
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.) <b>Note:</b> 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.
<b>Tag tabs: Real Time, OnBoard Alarm, Control, and Reset</b>	
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.

Field Name/Valid Entries	Comments
IEC Tag Name/Display only	Tag name that conforms to IEC61850 standard. See <a href="#">Tag Types on page 83</a> 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. <a href="#">Tag Types on page 83</a> for a detailed description of address construction.
Normally Closed (Control tags only)/ Check the box to invert the functionality of the control. See description.	<p>For a control with one command, writing a 1 to the tag will cause the command to occur. (This option is greyed out.)</p> <p>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.)</p> <p>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.)</p>
Edit Addr/Click to display Edit Address screen. (Real Time and Onboard Alarm only)	<p>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).</p> <p>See <a href="#">Edit Tag Addresses on page 52</a>. for detailed information.</p>
Register 1/Display only (Real Time tags only)	<p>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.</p>
Num Registers/Display only (Real Time tags only)	Displays the number of registers used by this tag.
Formatting/Select the format type from the drop-down list (Real Time tags only)	<p>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 <a href="#">Edit Tag Addresses on page 52</a>.</p>
Scaling Register/View or enter the register number (Real Time tags only)	<p>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.</p>
Functional Address/Display only (Real Time, Onboard Alarm, Control, and Reset tags)	<p>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.</p> <p>Note: Functional addressing is described in <a href="#">Appendix 4: Glossary on page 272</a>.</p>
Edit Functional Address/Add the code for the address	<p>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 I/O Device Manager.</p> <p>A simple example: Address = "T:MV;m:" + (startingpoint + 1005).ToString() + ";L:P:22"</p> <p>You would then define "startingpoint" when instantiating the profile in the I/O Device Manager.</p>

Field Name/Valid Entries	Comments
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 <a href="#">Set Up Custom Tags on page 49</a> 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 <a href="#">Set Up Custom Tags on page 49</a> for more information.

### Add, Edit, or Delete a Device Type

Use the Add/Edit Device Type screen to begin adding, editing, or deleting a device type from the system. To view this screen, click the Define Device Type Tags tab; then click Add/Edit, to the right of the Device Type Name field.

The following table describes the parts of the Add/Edit Device Type screen.

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

Field Name	Valid Entries	Comments
Create New	Click one of the radio buttons to select the action you want to take.	Click to add a device type that is not based on an existing type.
Create From		Click to copy an existing device type.
Edit Existing		Click to edit an unlocked device type.
Delete Existing		Click to delete an unlocked device type that is not associated with a profile.
Device Type (to Create From/to Edit/ to Delete)	select type	Select the device type that you want to create from, edit, or delete.
Copy Addressing		Active when you choose Create From. Check this box if to copy the addressing of the "from" device. This gives each tag in the new device type the same address string as the matching tag in the "from" device.
Device Type Name	Type or select the name: maximum 32 characters, do not use \ / : * ? < >	If creating a device type, type the name. If editing a device type, the device type that was selected for editing displays here. You can change the name here.
Lock this Device Type	Check to lock device, so that it cannot be edited.	This action cannot be undone. You cannot edit a locked device type. If it is a standard device type, you cannot edit or delete it. If you added the device type, you can delete it but not edit it.



Field Name	Valid Entries	Comments
Device Category	Choose the category for this device.	To create categories, see <a href="#">Print the .CSV File on page 47</a> . In addition to predefined categories, you can add custom categories. See <a href="#">Set Up Device Type Categories on page 47</a> 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.	Default options are Monitoring Device, PLC, or Protection Device. Depending on the device you select at the top of the page, this field is filled in for you.  As with categories, subcategories are created in the <a href="#">Print the .CSV File</a> screen.
Driver	Select the driver for the device type.	Predefined drivers are created for all PowerLogic compatible devices, though you may need to use these drivers for multiple device types. For example, you would use the CM4000 driver for a CM3000.  Use the Generic Power Device driver for third-party devices. The Controllable Device driver is currently not used. Use Generic JBus Device driver for JBus devices.
Display Associated Profiles	(Active only in Edit mode) Click to display a list of profiles that are associated with the selected device type.	This list is used when you want to delete a device type that is associated with a profile. In Edit mode, select the device type you want to delete; then click this button. Note the profile(s) associated with the device type.  Before you can delete the device type, go to the Add/Edit Device Profile screen, locate each profile in the list. You cannot save an empty profile, so you must either select another device type for it or delete the profile. Then you can delete the device type.
IEC Tags	n/a	This list includes all tags that have been added to the system, standard tags as well as custom tags that you have added. Tags are listed in their groups (such as 100ms, Onboard Alarm, Power Factors).
Selected Tags	Select tags from Tags; click the right arrow to move them to this box.	You can move single tags or entire tag groups. They must be moved one at a time (cannot shift+click to select).  Note: You cannot deselect tags for a device type if that device is associated with a device profile.
Add/Edit Custom Tags	Click to begin adding a custom tag.	Live when creating or editing a tag. Opens the Add/Edit Custom Tags screen. See <a href="#">Add Custom Tags and Tag Addresses on page 49</a> 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 49](#)).  
Real Time Tags sub-tab: you can edit the address (see [Edit Tag Addresses on page 52](#)) and choose a different format.
- From the Define Device Type Tags tab, select the device type you want to edit, then click Add/Edit. Follow through the screens to edit additional information:  
In the Device Type Options box, click Edit Existing.  
Click the Device Type to Edit list to display the Select Device box. Select the device type that you want to edit.  
You can change the device type name, category, subcategory, and driver. You can also lock the device.  
Select tags and tag groups and move them into or out of the Selected Tags list.  
If a device type is associated with a device profile, you cannot deselect tags.  
After all of the appropriate changes are made, click Save to save your current settings.  
To create additional custom tags, click Add/Edit Custom Tags; otherwise, click Save & Exit to save your settings and close the window.  
If you add a tag to a group that is already included in a device type, you must then individually add the tag to that device type.

### Delete a Device Type

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

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

To delete a device type:

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

### Assign Tags to Generic I/O Points

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

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

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

4. Edit the address of the Current A tag to match the address of Channel 1.

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

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

## Print the .CSV File

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

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

Follow these steps to create the CSV file:

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

## Set Up Device Type Categories

Use the Set Up Device Type Categories screen to add, edit, and delete categories. These categories are used in the I/O Device Manager 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.

Field Name	Valid Entries	Comments
	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

Use this feature to add variables to addressing. You can re-use a variable by copying/pasting parts of it into other addresses, then making changes to the code for use in other tags. You will be prompted for these variables in the I/O Device Manager.

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 I/O Device Manager 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 PowerSCADA Studio; 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 I/O Device Manager.
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 52](#)

[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 <a href="#">Delete a Custom Tag on page 52</a> for instructions on using this button.
Tag Name	Type the new tag name; or type the changed name for a tag you are editing.	Maximum 32 characters; can include any alpha or numeric character, as well underscore (_) and backslash (\). Must begin with either an alpha character or underscore.
Display Name	Type the name that you want to display when selecting the tag and in other displays.	You might use this field for additional information on the Add/Edit Custom Tags screen. For example, you could describe the data that it logs. It does not display anywhere else in the system.
Group	Select the group.	Includes all of the real-time groups (such as 100ms, controls, currents) plus onboard alarms, resets, and controls.
Data Type	Select the data type.	These are PowerSCADA Expert tag data types. They affect the logic codes that are available for display in the Edit Address screen. See <a href="#">Logic Codes on page 87</a> 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.
PowerSCADA Studio Format	Select the numerical format.	This is used for display purposes in PowerSCADA Expert graphics pages. It determines where the decimal displays.

Field Name	Valid Entries	Comments
		Choose the reporting format, to be used in PowerSCADA Expert, from ## to #0.#####. For example, if you select #.##, the number 8.12579 would be displayed as 8.12.
Polling Priority	Low, Normal, or High	Indicates the level of priority PowerSCADA Expert uses when reading data from devices. Note: In the address field, a priority of 1 = High, 2 = Medium, 3 = Low.
Alarm On Text	For onboard alarms only: enter the text for when the alarm is On.	This text displays on the Create Device Profiles tab for the onboard alarm tag, when it is selected for the device type in the profile. It also displays in the Alarm Log.
Alarm Off Text	For onboard alarms only: enter the text for when the alarm is Off.	
Display 'Advanced' filter selections	Check to display additional filter options in the Real Time Filter and Alarm Filter tabs	Displays several additional filter options on the two "Filter" tabs. These options will be useful in the future for reporting purposes.
<p>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.</p> <p>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.</p>		
<b>Real Time Filters tab</b> (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.
<b>Alarm Filters tab</b> (dropdown lists are expanded when "Display 'Advanced' filter selections" is checked)		
Categorization	Select the alarm category	Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting.
Alarm Type	Select the alarm type.	Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting.
Alarm Group	Select the group.	Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting.
Subcategorization	Select a subcategory.	Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting.
Alarm Level	Select the severity level of the alarm.	Used for filtering and sorting alarm data. Also metadata for future use in statistical reporting.

### Edit a Custom Tag

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

1. Open the Add/Edit Custom Tags screen (from the Add/Edit Device Type screen, click Add/Edit Custom Tags).
2. In the Custom Tag Options box, click Edit Existing.

3. You can change any of the tag attributes. (This does not change the tag's assignment status; if it is selected for a device type, it does not move back to the IEC Tags list.)
4. Click Save to save changes, or click Save & Exit to save changes and close the screen.

### Delete a Custom Tag

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

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

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

Continue deleting the tag:

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

### Edit Tag Addresses

Use the Edit Address screen to edit the attributes of a single tag address. If a device type is locked, you cannot edit any of its tag addresses; they will be greyed out. A thorough discussion of IEC61850 tags and their construction is included in [Tag Types on page 83](#) and [Logic Codes on page 87](#).

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

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

## Real-Time Tag Addresses

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

Field Name	Entry	Comments
Data Type	For display only	You can edit this field in the Add/Edit Custom Tags screen.
Priority	High, Normal, or Low Logic Code:	You can edit this field either here or in the Add/Edit Tag screen.
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 <a href="#">Logic Codes on page 87</a> .



Field Name	Entry	Comments
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 <sup>-2</sup> which becomes 1.23.
Offset	$y = ,x + b$	y = the final value reported by PLSCADA b = the offset m = the conversion factor x = the original value in the meter b = rarely used, mainly in temperature conversion The offset is added to the final value (after the conversion factor is applied).
Register 1-4	Enter the register number.	Be aware of whether you chose hexadecimal or decimal. Use the same format here.
Bitmask for Register 1-4	For digital input/output tags: Set the bits to 1 or 0 to match the pattern for "True" in the device register.	When all bits match exactly the pattern in the register, the status is True. When any one bit does not match the pattern in the register, the status is False. Note: On PM8s and CM4s, there is a device-specific format, DIgIn and DIgOut. In each case, you must first specify the indicator register (which becomes the first register). The second register will have the mask.
Invert Result	Check this box to invert.	Will turn False to True or vice versa; typically used for Normally Open or Normally Closed.

## Onboard Alarm Tag Addresses

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

Field Name	Entry	Comments
Tag Name	For display only	This is the tag name, which cannot be changed.
File Number	Select the number.	This is the file number for the alarm file on the device. (Sepam has no file number; enter 0.)
Module	Select the module.	Choose the type of module in which the tag is used. Used for Micrologic at this time.
Unique ID	Choose the identifier.	This unique identifier must be used to ensure that alarms will annunciate correctly. For CM4, PM8, PM5000, and Micrologic, the unique ID must be decimal. For Sepam, the unique ID is the coil bit address that indicates the alarm; it must be in hexadecimal.
Hexadecimal	check box	Check this box if you want to display the ID in hexadecimal, rather than decimal.
Has Unique Sub ID	check box	Check if this tag has a unique sub-identifier (Micrologic, CM4000, PM800, and PM5000 devices).
Unique Sub ID	Enter the Sub ID.	Enter the unique sub-identifier. Active only if Unique Sub ID box is checked.

## Reset Tag Addresses

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

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

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

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

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

Box Name	Field Name	Comments
Tag Information	Command Type	The Command Type and Command to Edit are already selected.
	Command to Edit	
Data Information box	Data Type: for display only	You can edit this field in the Add/Edit Custom Tags screen.
	Priority: High (default)	Cannot be edited.
	Logic Code: Select the logic code for this tag.	Choose the appropriate logic code for this tag. See <a href="#">Logic Codes on page 87</a> .
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.

Box Name	Field Name	Comments
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 <a href="#">Logic Codes on page 87</a> .

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. <b>Note:</b> 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 Citectscada help file. For detailed information, see **Variable Tag Properties** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## Create Device Profiles

Use the Create Device Profiles screens to view and edit profiles for individual devices. Profiles are predefined for the standard devices; you will mostly use this feature to add third-party device profiles. Before you create profiles, you need to be sure that all of the tags and device types that you need are created (see [Define Device Types and Tags on page 41](#)). 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 57](#)

[View Device Profiles on page 57](#)

[Add Edit or Delete Device Profile on page 59](#)

[Select Trend Intervals on page 61](#)

[Set Up Trend Intervals on page 61](#)

[Edit IEC 61850 Datasets on page 62](#)

[Edit IEC 61850 Report Control Blocks on page 63](#)

[IEC 61850 System Setup Workflow on page 25](#)

[Create a Composite Device Type on page 65](#)

[Create Data Concentrator Device on page 66](#)

[G3200 Device Setup on page 67](#)

[DNP3 Protocol Support on page 68](#)

## Enable Waveforms

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

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

As device information is polled and received by PowerSCADA Expert, the waveform becomes available for viewing. See *The Alarm Log* in [View the Alarms/Events Page on page 172](#) 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 57](#) for more information). To begin making changes, click Add/Edit. The following table describes the fields on this tab. The tags listed assume that Advanced Properties has been checked. Not all elements appear on every sub-tab. Detailed instructions are after the table.

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

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

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

## Add Edit or Delete Device Profile

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

### Add a Profile

1. Open the Create Device Profiles tab (from the Create Device Profiles tab, click *Add/Edit*).
2. In the Profile Options box, click Create New or Create From.  
If you are creating from another device profile, choose it from the Device Profile to Create From drop-down menu.
3. Click Next to make the name and description fields live.
4. Type a unique Device Profile Name, maximum 32 characters; do not use \ / : \* ? < > |
5. If you want to lock this profile, preventing anyone from editing it, check the Lock this Device Profile box. This action cannot be undone. If you wish to edit a locked profile, you must use the Create From option to add a new one, then delete the locked one.
6. (optional) Type a device description. This will display as a tool tip in later screens.
7. Click Next again to make the remaining fields live.
8. From the Available Devices list, highlight the first device or device group (Protection, Monitoring, Composite) to be included in this profile. Click the right arrow button to move it to the Selected Devices box. You must select and move devices or device groups one at a time (no shift+click to select multiples).
9. If you will want to import this project into another instance of the Profile Editor, see [Add Project Parameters on page 72](#).
10. When you have all of the devices you want, click Next.
11. From the Device Type Tags list on the left, select the tags you want to include in this profile. You can select entire tag groups or individual tags from a group; but you must select them one at a time.
12. After each addition, the tag or tag group displays in the Selected tags box. You can override any tag name (typically for generic I/O devices with multiple tags, such as inputs, for which names alone would not be intuitive in runtime. To override a tag, select it, then click Override Tag Name. Choose the tag you want. Click OK. The new tag will correctly display the value of the original tag, but will take the appearance of the override tag (such as description, metadata).
13. The final column, Is Device Tag, displays only for composite devices. Check this box to tie a tag back to its actual physical device. For example, if the same tag is in three devices, and you set PC-based alarms for each device, you need to be able to determine which device has a problem in runtime. To prevent confusion, check Is Device Tag to cause PowerSCADA Expert to report the tag for its physical device, rather than the composite device.
14. When you have selected all tags, click Next.  
  
NOTE: If you have duplicate tags from multiple devices, you need to resolve this by using an override for one of the tags.
15. On the next page, choose whether each tag will have a PC-based alarm and/or trend associated with it. Click Finish.

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

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

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

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

For instructions on changing the “short” scan interval settings, see [Trend Tag Scan Intervals on page 163](#).

To change a trend interval for a tag, see [Select Trend Intervals on page 61](#) To add additional trend tags, see [Set Up Trend Intervals on page 61](#).

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 62](#) and [Edit IEC 61850 Report Control Blocks on page 63](#) 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 61](#)).
  - 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 49](#)).
  - Onboard Alarms sub-tab: enable waveform capture for on-board alarms (see [Enable Waveforms for Onboard Alarms on page 156](#) 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 49](#) for more information).
  - Parameters sub-tab: Edit parameters for IEC 61850 driver parameters (see [Edit Driver Parameters on page 19](#) 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 61](#).

To change a trend interval, follow these steps:

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

## Set Up Trend Intervals

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

### To add a trend interval:

1. Access the screen: Click Settings > Set Up Trend Definitions.
2. From the Set Up Trend Definitions screen:
  - a. Click New to begin adding a new trend
  - b. Select a trend, then click Copy to create a new trend from an existing trend.
3. Type a name: must begin with either an alpha character (A-Z or a-z) or the underscore character (\_). Any following characters must be either alpha characters (A-Z or a-z), digit characters (0 - 9), backslash characters (\), or underscore characters (\_).
4. Type the appropriate information in the following fields. For detailed information, see **Trend Tag Properties** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).
5. Click OK to save the new trend definition, which displays in the left-hand box.

### To edit a trend interval:

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

### To delete a trend interval:

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

## Edit IEC 61850 Datasets

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

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

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

## Create a New DataSet

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

The fields on the right side of the screen become live.

2. Type a name and description for the new dataset. These are free-form fields, but they must comply with IEC 61850 standards.
3. Choose the appropriate logical device, then choose the logical node for that device.
4. Choose the functional constraint for the content. This will filter the display of device type objects/topics in the box below.

When you choose All, you must then choose an object that already has a functional constraint in it. If you choose a specific constraint, the list of available objects is filtered to display only those that include that constraint.

5. From the Device Type Objects, choose the appropriate objects for this profile.
6. Click OK.

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

## Create a DataSet from an Existing DataSet

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

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

The new name displays under the Device Profile List.

## Copy a DataSet to a Device Type

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

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

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

## Edit and Delete DataSets

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

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

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

## Edit IEC 61850 Report Control Blocks

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

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

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

### Create a New Report Control Block

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

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

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

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

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

The new name displays under the Device Profile List.

### Copy a Report Control Block to a Device Type

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

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

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

### Edit and Delete Report Control Blocks

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

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

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

## Create Third Party Modbus Device Type

You can create a third party Modbus device and add it to your PowerSCADA Expert project. Follow the steps below:

1. Find the Device Modbus Reference. This should be included in a document from the manufacturer for the device you want to add.
2. Familiarize yourself with the manner that the Modbus device specification.
3. Verify the PowerSCADA Expert supports the device: The following table lists allowed values for each data type:

Data Type	Variable	Size	Allowed Values
String	string	256 bytes (maximum)	ASCII (null terminated)
Digital	digital	1 bit or 1 byte	0 or 1
Long	long integer	4 bytes	-2,147,483,648 to 2,147,483,647
Real	floating point	4 bytes	-3.4E38 to 3.4E38

4. Verify that the tags you want to use are compliant with PowerSCADA Expert. To ensure that data is reported for reporting, LiveView tables, and breaker graphics. Refer to the Common Data Model (CDM), which is located in C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\v8.2\Applications\AppServices\bin..
  - a. Find the best fit tags: From the Profile Editor tag library, find the tag that comes closest to the quantity you want to measure.
  - b. Verify the tag you have chosen by comparing it with the CDM. The DisplayName
5. Create the device type in the Profile Editor: Use PwrModbus as the driver.
6. Select the appropriate tags (CDM)
7. Configure the Modbus tags: Continuing on the Define Device Type Tags tab, edit the tag addresses to map them to the Modbus register of the device (these tags will be red). You can locate instructions on editing addresses in the PowerSCADA Expert\help file.
8. Create the device profile: Click Add/Edit to launch the Add/Edit Device Profile window. Create the new profile and choose the device(s) that you want.

9. On the next screen, move the tags into the Selected Tags pane. Select Trend for all tags that require it.
10. Continue with setting up the project and exporting as you do with other device profiles.

## Create a Composite Device Type

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

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

### Cyber Sciences Sequence of Events Recorders (SER)

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

Follow these steps to create the composite device type:

1. From the Create Device Profiles tab, click Add/Edit.
2. At the Add/Edit Device Profile screen, choose whether you are creating a new device or creating from an existing device. If you are creating from a device type, select it. Click Next.
3. Still on the Ad/Edit Device Profile screen, give the composite device type a name. Optionally, add a description (which will become a tool tip display in later screens). Click Next.
4. Choose the device types to be in the composite. Click Next.  
The Add/Edit Device Profile displays with only device type tags available for selection.
5. Add the tags you need for each device type listed on the left. To add all of the tags for a device type, highlight the device type name and click the right green arrow.  
You may find, especially when dealing with generic I/O, that the tag name is not descriptive enough to determine what it is when reading data in runtime mode. Thus, you may want to override the generic name with something more meaningful.  
For example, a device may have ten inputs: Ind1, Ind2, Ind3, and so on. Using those names, you have no idea what each input is reading. If you override the tag, the tag's value will still come from the original tag (it still keeps the addressing from the device); however the tag's appearance (name, metadata, display name) will be taken from the new tag.
6. To override a tag:
  - a. Highlight the tag, then click Override Tag Name.
  - b. From the Select Tag window, choose the tag you want. If necessary, enter a search term, then click Search to display related tags.
  - c. Choose the tag, then click OK.

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

you would check the *Is Device Tag*, which will cause PowerSCADA Expert to report this tag for its physical device.

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

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

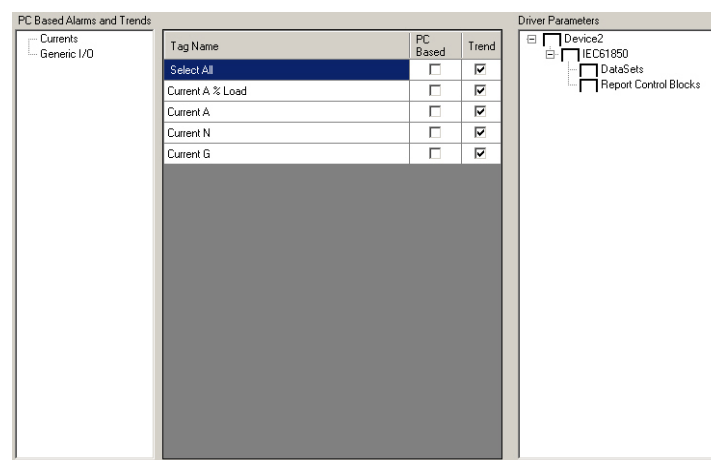
When the profile is added to the project, historical trends are added to the Trend Tags file. Logging will automatically begin when the tag is added to the project.

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

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

For instructions on changing the “short” scan interval settings, see [Trend Tag Scan Intervals on page 163](#).

10. The Driver Parameters box allows you to specify certain parameters to be attached to device profiles. Currently used in IEC61850 devices, the available parameters will automatically populate this box. See the illustration below for an example.



In this example, Device 2 has two parameters, DataSets and Report Control Blocks.

11. Check the parameter(s) that you want to include in this profile.  
To edit, this parameter, return to the Create Device Profiles tab, and click the Parameters sub-tab. See [Edit IEC 61850 Datasets on page 62](#) and [Edit IEC 61850 Report Control Blocks on page 63](#) for information on editing these two parameters.
12. Check the Close Wizard box, and click Finish to return to Create Device Profiles tab/ Or, leave it unchecked, and click Finish to return to the Add/Edit Device Profile screen.

## Create Data Concentrator Device

When you create a data concentrator and/or a data-concentrated devices in the Profile Editor, all of the related devices must use the same protocol. Examples of data

concentrators are PLCs that use inputs from various devices or an RTU that concentrates data from multiple devices.

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

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

## G3200 Device Setup

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

### For use with multiple devices

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

### In the Profile Editor

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

### In CET850

Create the CID file for the G3200 gateway.

### In PowerSCADA Expert

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

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

### For use with a single device

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

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

## In the Profile Editor

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

## In CET850

Create the CID file for the G3200 gateway.

## In PowerSCADA Expert

1. Open the I/O Device Manager.
2. Add the device.
  - a. From the Enter instance information screen, select the BRCBs that you need.
  - b. Change the logical device as needed.
  - c. Select the CID file you created in CET850.
  - d. Complete the remaining steps in the I/O Device Manager.

## DNP3 Protocol Support

You can create device types and profiles that use the DNP3 protocol. See [Add a DNP3\\_TCP Device on page 123](#) or [Add a Serial Device to a Project on page 120](#) 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 69](#)
- [The Set Up Projects Tab on page 70](#)
- [Add, Edit, or Delete a Project on page 70](#)
- [Create a Composite Device Type on page 65](#)
- [Edit and Delete Information in a Project on page 72](#)
- [Customize Tag Names on page 72](#)

## 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 I/O Device Manager. If PowerSCADA Expert is installed and the corresponding PowerSCADA Expert project has been created, this also copies the file that is used by the Device Creation Wizard to the PowerSCADA Expert project.

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

### Typical Workflow

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

1. Open the Add/Edit Project screen.
2. Add a new project, or copy and edit an existing project.
3. Select the device profiles that you want to use for this installation.
4. If a device profile has multiple drivers, choose the driver, and determine whether the individual device types will use functional addresses and act as data concentrators.
5. Save the project and close the Add/Edit Project screen.
6. To customize tag names: From the Set Up Project tab, click the Customize Tag Names sub-tab. Change the name of any tag. For example, the customer might need "Current A" to read "Current Phase A." The customized tag name will be used in all device profiles in the project for which you have created the customized tag. For this change to be in the PowerSCADA Expert project: you need to delete the device profile from that project and then re-export it.
7. To add optional project information: From the Set Up Project tab, click the Project Parameters sub-tab. You can add optional information that will help verify or identify this project later. You could, for example, add the version or the creator's name. This information will be available when you import this project at a later date.
8. To refresh tags: From the Set Up Project tab, click the Selected Device Profiles sub-tab. Click the Refresh Tags button for any profile. You are prompted to confirm that

you want to update changes you have made to this tag for the selected profile. For this change to be in the PowerSCADA Expert project: you need to remove the device profile from that project and then re-add it.

9. Click Export Project to create an Equipment.Profiles file of all of the profiles included in the project. To view Equipment.Profiles, click the folder button, to the right of the Export button:



## The Set Up Projects Tab

Use the Set Up Project tab to begin adding, editing, or deleting projects. A project includes all of the tags that belong to the device profiles that you have created and added to the project. From this screen you also export individual projects to the .XML file format, which you can add via the I/O Device Manager.

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 72](#).
- 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 72](#).

To add or edit project information, click Add/Edit. The Add/Edit Project screen displays. See [Add, Edit, or Delete a Project on page 70](#) 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 73](#) 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 I/O Device Manager).

### 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 37](#) 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 40](#) 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 272](#).)  
Click  to move the device profile to the Selected Device Profiles box.  
If the Select Profile Drivers screen displays, one of the following is true.
  - You did not check the "Add As Default" box for a device type, so the system does not know how to use the functional address/data concentrator option. Check the appropriate box to turn the related option "on."
  - At least one of the device types in this profile includes multiple drivers. For each multiple-driver device type listed, choose the driver that you want to use in this project. Additionally, you can check either of the Functional Address or Data Is Concentrated boxes to enable those features.
10. Give the device type a Configured Name. This name might indicate its status (which driver it uses, whether it has a functional address, and so on) in future project references.
11. When all profiles are added, click Save to save the changes, or click Save & Exit to save changes and close the screen.

### Edit a Project

You can only edit projects that are unlocked.

To edit a project:

1. Open the Add/Edit Project tab (from the Set Up Project tab, click Add/Edit).

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

### Delete a Project

You can only delete unlocked projects.

To delete a project:

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

## Edit and Delete Information in a Project

After you have exported a project to a PowerSCADA Expert project, you still need to use the Device Creation Wizard to add system information to the PowerSCADA Expert project. See [Before You Use the I/O Device Manager on page 117](#) 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 I/O Device Manager.

## 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 73](#).
- 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 75](#).

- 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 74](#).
- Import a project from another instance of the Profile Editor or from an IEC61850 file.
- Import SCL files. You can import from the profile data of IEC61850-compliant devices and create device types. These files can be exported as an IID profile or as a PowerSCADA Expert profile.
- Import ICD files. You can import either functional constraints or report control blocks.

The import process works the same for each type of import. The only exception is that you cannot import profiles when you are importing SCL files. See [Import Files into the Profile Editor on page 76](#).

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 81](#).

## 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. From the PowerSCADA Studio Projects tab, add a new project. Be sure that Template Resolution is SXGA.  
If you have questions about any of the fields, click Help.
2. Add your project to the Profile Editor, ensuring that the name matches exactly the one that you added in PowerSCADA Expert (to ensure that it correctly exports to its matching project).

### To export a Profile Editor project to the PowerSCADA Expert project:

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

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

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

- Each Project file, used by the Profile Editor, is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v8.2\Projects.
- Each I/O Device Manager profile file is stored in Documents and Settings\All Users\Application Data\Schneider Electric\Profile Editor\v8.2\WizardProfiles\[project name]. A single file for each included profile.
- The Equipment.Profiles file (contains all of the I/O Device Manager profile information and the base profile information used by the I/O Device Manager) is stored in ProgramData\Schneider Electric\PowerSCADA Expert\v8.2\User\[project name].

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 I/O Device Manager).

5. 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.
  - a. Click the folder icon beside the Export link:



- b. Copy the file (Equipment.profiles) that displays and move it to a portable drive.
2. On the PowerSCADA Expert server computer, paste Equipment.profiles in this location:

[Drive Letter]:\Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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).

3. Use the I/O Device Manager 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 158](#).

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 72](#) and [Add Project Parameters on page 72](#) for the information you need to make the changes that you want.
5. Click Export Project.

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

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

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

C:\Program Data\Schneider Electric\PowerSCADA Expert\v8.2\Applications\Profile Editor\Projects\YourProjectName.pls.

## SCL Export

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

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

The only ways you can edit these files are:

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

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

### Exporting the File

Follow these steps to export IID files:

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

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

The Export Summary displays with the results of the export. When the export displays under the Success topic, the listed files were exported. When the export displays under the Warnings topic, the reason that the export did not succeed is listed for the device types shown.

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

[Project Drive]\Program Data\  
SchneiderElectric\PowerSCADA Expert\v8.2\Applications\Profile  
Editor\WizardProfiles\<<project name>\SCL\_Export\sclFileName.iid).

## Import Files into the Profile Editor

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

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


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

Follow these steps to import data into the Profile Editor:

1. Before you begin, consider the source of the information you want to import. We strongly recommend that you use a master PC from which you draw this information. This will ensure that you are using a single source. Also, back up your data folder before you start. This gives you data to revert to, in case you accidentally lose data.

2. Note the location of the project file or other file (SCL, CID) that you want to import.

3. From the Profile Editor, click File > Import.

4. At the Import File Selection window, click browse (  ), then navigate to the location of the file you want to import.

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

Use this information to verify that you are about to import the files that you want.

5. If desired, select an import template from the drop down list. (To create a template, see [Creating a New Template During Import in Using Import Templates on page 81](#).) If you select a template, the import will accept default properties from the template. For example, if the template has alarm settings from a device, and you are importing tags for that device, the import will use those alarm settings.

6. When you locate the desired file, click Open to choose it. Then click Start Import.

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

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

If you are importing IEC 61850 data, the Import Filter screen displays. Use this screen to perform an initial filter on functional constraints or on report control blocks. See [Import Reconciliation Screen on page 78](#) for more information.

7. Make your selections, then click Continue.

8. When the Import Reconciliation screen displays, you can begin the process of matching or rejecting individual tags. See [Import Reconciliation Screen on page 78](#) for a description of the parts of this screen.

9. On the Import Reconciliation screen, click an item in middle pane. Respond to the item according to your preference for it. You must set the status first for units, then tags, and finally the device type.



10. After you match or ignore all items in the import list, the Complete Import button becomes live. Click Complete Import.

After the import is saved, the Save Import Template dialog displays. See [Using Import Templates on page 81](#) for instructions on creating, using, and deleting import templates.

### Import SCL Files

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

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

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

You can save the information in one of two ways:

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

## Import Filter Screen

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

### Functional Constraints

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

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

### Report Control Blocks

1. Click the Report Control Block button.

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

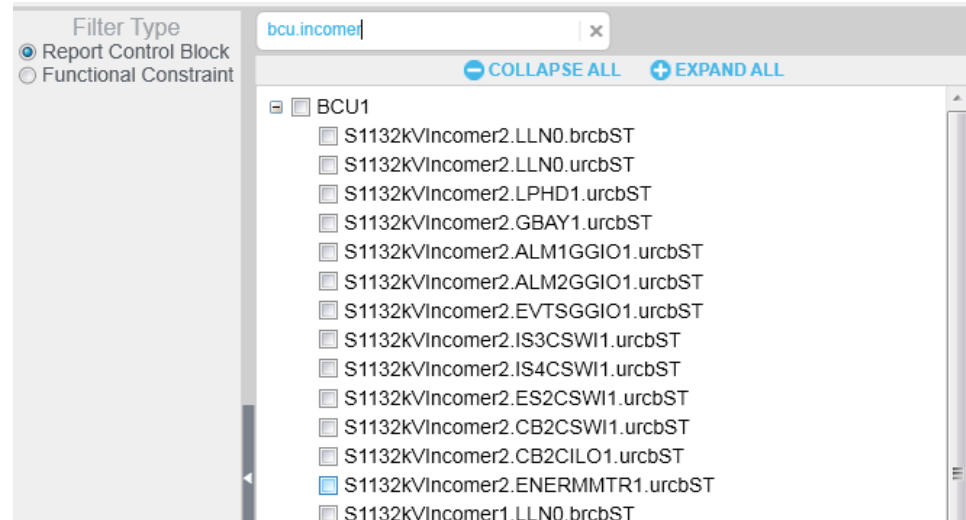
The right-hand column displays the IEDs/report control blocks that you have selected.

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

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

*bcu.incomer*

The result would look something like this:

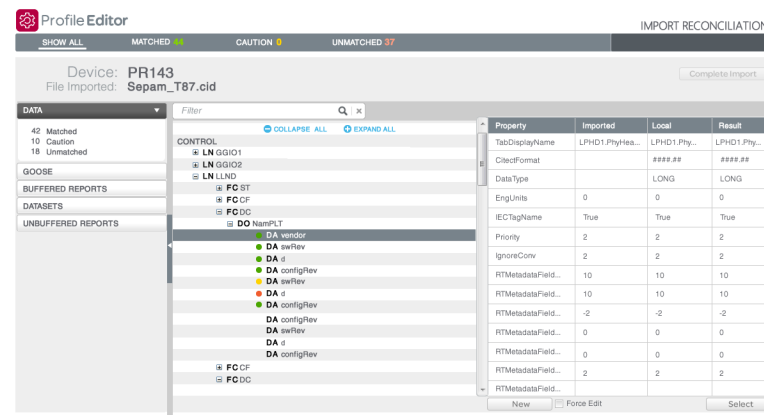
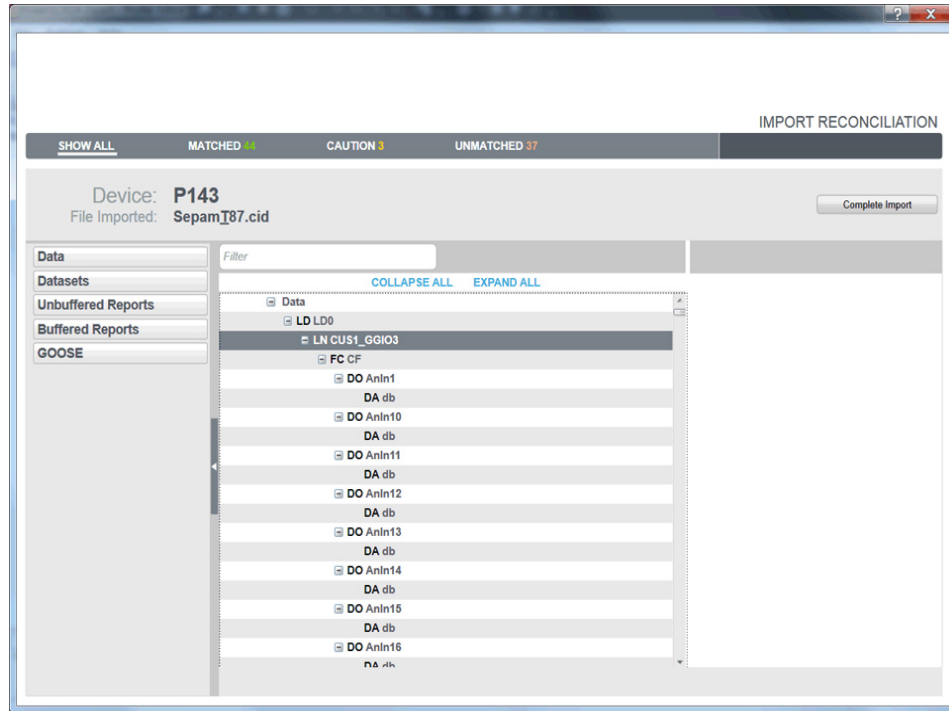


When you have selected either the functional constraints or report control blocks, click Continue. The data is filtered on the last filter option that you chose (you cannot combine filters).

The Import Reconciliation screen displays. See [Import Reconciliation Screen on page 78](#) for help finishing the import.

## Import Reconciliation Screen

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



The screen is divided into three panes:

### Left-hand pane:

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

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

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

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

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

**Middle pane:**

This pane shows a tree view with data.

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


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


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


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

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

The bullets indicate:

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

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

 : no match; item does not match any local item

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

**Re-match Items within a Logical Node**

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

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

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

ST.PhyHealth.q

ST.PhyHealth.t

ST.Proxy.q

ST.Proxy.stVal

ST.Proxy.t

**Right-hand pane:**

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

- Property: The property for which the other columns provide definitions.
- Imported: The value of the item in the import file.

- **Local:** The closest local match for the imported item.
- **Result:** The item as it will be added in this import; by default, this item is inherited from the local status.

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

**Force Edit:** Check Force Edit to display a screen that lets you edit the item's information. You can make changes to an item, even though it may be an exact match with a local item. This new information will be applied to the item after you complete the import.

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

### Complete the Import

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

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

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

## Using Import Templates

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

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

## Creating a New Template During Import

To create a new template:

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

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

## Applying a Template During Import

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

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

To apply an existing template:

1. From the Set Up Projects tab, select a project for which you want to import data.
2. Click File > Import, then choose the file (.pls or .icd) that you want to import.
3. From the Import Template drop down list, choose the template you want to use. This is just a starting point for this import to make it quicker to match items. You will apply the template in step 7.
4. Click Start Import.  
After the import completes, the Import Reconciliation screen displays. The list in the left-hand pane should have some exact and partial matches.
5. As you work through the items, you must either designate that each a match or ignored.
6. When all items are completed, click Complete Import.
7. At the Save Import Template dialog, click No to import without applying a template. Or click Yes to either save a new template or edit the one you chose in step 3:
  - To create a new template for this import, click New, then type an Import Template Name.
  - To edit a template, click Edit, then select the template from the drop down menu. This will edit the template by adding the changes you made during matching. This cannot be undone after you click OK.
8. Click OK.

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

## Deleting a Template

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

To do this:

1. Click Settings > Remove Import Templates.

- At the Import Templates dialog, select the template you want to delete, then click Delete.

The template is deleted.

## Tag Types

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

In this section, you can learn about:

[IEC 61850 Tag Construction on page 83](#)

[Define an Enumeration on page 84](#)

[Format Code Definitions on page 85](#)

[Logic Codes on page 87](#)

[Block Writes on page 98](#)

## IEC 61850 Tag Construction

IEC 61850 tags are flexible, which allows them to specify how functions are implemented in devices. The IEC 61850 tag was developed for medium-voltage and high-voltage applications, such as monitoring, control, and substation automation.

Some of our devices include data and functionality that are not yet covered by IEC 61850. For these devices, the general IEC61850 formatting was followed when creating tags.

If you are writing Cicode (see [Use Cicode to Customize a Project on page 159](#)). You will need to know the IEC 61850 tag name that you added to the device profile for that device. You can print the .csv file to view tag names (see [Print the .CSV File on page 47](#)). 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)

Category Name	Description
Pxxx	protection; e.g., PDIF (differential), PIOC (instantaneous overcurrent or rate of rise.), PDIS (distance), PTOV (time-overvoltage)
Rxxx	protection related; e.g., RREC (auto reclosing), RDRE (disturbance)
Sxxx	sensors, monitoring; e.g., SARC (arcs), SPDC (partial discharge)
Txxx	instrument transformer; e.g., TCTR (current), TVTR (voltage)
Xxxx	switchgear; e.g., XCBR (circuit breaker), XCSW (switch)
Zxxx	other equipment; e.g., ZCAP (cap control), ZMOT (motor)

The following example illustrates the IEC 61850 tag for current A:

EquipmentName\MMXU1\A\PhsA

where:

M = the category

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

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

A = the data object, current

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

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

## Define an Enumeration

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

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

Bit x   Bit y	Status	Circuit Breaker Position	Returned Value
0 0	Indeterminate	Circuit breaker is neither open nor closed	0
0 1	Open	Circuit breaker is open.	1
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 52](#) 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 239](#)), or create a custom genie (see [Create New Genies on page 115](#)).

## 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 87](#) 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	

Attributes	Modifiers	Comments
	c = output coil (writable only) in decimal l = input register (read only) in hexadecimal i = input register (read only) in decimal	
Register Number Modifiers (register number from 1–4)	u## = ## registers are unsigned; ## is a decimal s## = ## registers are signed; ## is a decimal	After the modifier, there may be a number indicating scaling factor. See “V,” below in this table. Used for conversion to SI units, this number will be: RegisterValue x scale For SS and DS: there must be a 1U default; the modifier will be a bitmask: - The mask must use hex only, 16 bits/register - Attach the ones, then the zero mask, to the register; if you only have ones masks, just attach them - Only one register cases can be inverted. Add :I after the masks for inversion.
N (scale)	numerical entries; range is -10 to 10	N defines a constant scale; the logic code knows how to use it.
R (scale register)	the register number in decimal	R defines the holding register where the scale is held; the logic code knows how to use it.
E (priority)	single digit: 1, 2, or 3; default 2 is used if this is not included (1 = high, 2 = normal, 3 = low)	Defines the priority PowerSCADA Expert uses in processing data.
V (conversion factor)	Use scientific notation without the decimal.	Examples: 354E-3 = 0.354 354E1 = 3540 Will be multiplied before the value is returned.
L:P (logic code) Required	The number that is used comes from the Logic Codes table.	L:P is the logic code for PowerLogic. Other codes may follow, such as L:I for ION. For logic code descriptions, see <a href="#">Logic Codes on page 87</a>

### 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 <a href="#">Logic Codes on page 87</a> . Write Address format: T:SS;m:##.#:L:P:101 Example: C:NO;T:SS;m:1234:1;L:P101;T:SS;m:3456:1;L:P101		

## Logic Codes

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

Logic Code	IEC Type	PowerSCADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
READS						
Invalid Block (L:P:0)	IB	LONG	Up to 1,000 sequential registers	No	Generic only	Defines invalid blocks of memory in the device. The driver does not include these registers in block reads.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
Date / Time (L:P:1) (3 register)	UT	LONG	3 sequential registers	No	Generic – if it fits	<p><b>Register N:</b> High byte = Month 1–12 Low byte = Day 1–31</p> <p><b>Register N+1:</b> High byte = Year 0–199 (+1900) Low byte = Hour 0–23</p> <p><b>Register N+2:</b> High byte = minutes 0–59 Low byte = seconds 0–59</p>
Date / Time (L:P:2) (6 register)	UT	LONG	6 sequential registers	No	Generic - if it fits	<p><b>Register N:</b> Seconds 0–59</p> <p><b>Register N+1:</b> Minutes 0–59</p> <p><b>Register N+2:</b> Hours 0–23</p> <p><b>Register N+3:</b> Day 1–31</p> <p><b>Register N+4:</b> Month 1–12</p> <p><b>Register N+5:</b> Year 0–199 (+1900)</p>
Date / Time (L:P:3) (3 or 4 register –Circuit Monitor/ Power Meter)	UT	LONG	3 or 4 sequential registers	No	CM/PM	<p><b>Register N:</b> High byte = Month 1–12, Low byte = Day 1–31</p> <p><b>Register N+1:</b> High byte = Year 0–199 (+1900) Low byte = Hour 0–23</p> <p><b>Register N+2:</b> High byte = minutes 0–59 Low byte = seconds 0–59</p> <p><b>Register N+3:</b> msec = 0–999 (unused)</p>
Date / Time (L:P:4) (3 or 4 registers SEPAM)	UT	LONG	3 or 4 sequential registers	No	SEPAM	<p><b>Register N:</b> Bits 0–6 = Year: 0 –70 (2000– 2070) 71 – 99 (1971–1999)</p> <p><b>Register N+1:</b> Bits 8–11 = Month Bits 0–4 = Day</p> <p><b>Register N+2:</b> Bits 8–12 = Hour Bits 0–5 = Minutes</p> <p><b>Register N+3:</b> msec = 0–59,999 (seconds are ms/1000)</p>
Date/Time (L:P:5) 3-register Micrologic	UT	LONG	3 sequential registers	No	micro	<p><b>Register N:</b> High byte = Month 1–12, Low byte = Day 1–31</p> <p><b>Register N+1:</b> High byte = Year 0–69 (+2000), Year 70–99 (+1900) Low byte = Hour 0–23</p> <p><b>Register N+2:</b></p>

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						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	<b>Register N:</b> High byte = Month 1–12, Low byte = Day 1–31 <b>Register N+1:</b> High byte = Year 0–69 (+2000), Year 70–99 (+1900) Low byte = Hour 0–23 <b>Register N+2:</b> High byte = minutes 0–59 Low byte = seconds 0–59 <b>Register N+3:</b> msec = 0–999 (unused)
Date/Time (L:P:7) 3-register Argos	UT	LONG	3 sequential registers	No	Argos	The number of seconds since 01/01/2000 (00:00:00) register 1 = MSB register 2 = LSB register 3 = milliseconds
Date/Time (L:P:8) 4-register IEC 870-5-4	UT	LONG	4 sequential registers	No	generic	<b>Register N:</b> Bits 0–6 = Year: 0–127 (2000–2127) <b>Register N+1:</b> Bits 8–11 = Month Bits 0–4 = Day <b>Register N+2:</b> Bits 8–12 = Hour Bits 0–5 = Minutes <b>Register N+3:</b> msec = 0–59,999 (seconds are ms/1000)
Modulo 10k (L:P:10)	BC	STRING	Up to 4 registers	No	generic	Result is a string representation. Range is 0 to 9,999,999,999,999,999 Each register has a range of 0 to 9,999 Result is: $-R4 * 10,000^3 + R3 * 10,000^2 + R2 * 10,000 + R1$
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: $-R4 * 10,000^3 + R3 * 10,000^2 + R2 * 10,000 + R1$  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.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
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 $-(R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1)/1000$
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 $-(R4*10,000^3 + R3*10,000^2 + R2*10,000 + R1)/1000$ 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.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
PL Digital Output TF (L:P:25)	SS	DIGITAL	2 registers	No	CM/PM	Same as PL Digital Output SS except: Result is: 0 = false and 1 = true. This result can be inverted.
Status SS (L:P:26)	SS	LONG	Up to 4 registers	No	Generic	Each register is compared to a ones' mask. Optionally it can be compared to a zeros' mask. (Use the Edit Address screen in the Profile Editor to create masks for the user.) Result is: 0 = off and 1 = on. If there is only one register, the result can be inverted.
Status OR SS (L:P:226)	SS	LONG	2 to 4 registers	No	Generic	Each register is compared to a ones' mask. These results are OR'ed together. Optionally, it can be compared to a zeros' mask. (Use the Edit Address screen in the Profile Editor to create masks for the user.) Result is: 0 = off and 1 = on.
Status DS (L:P:27)	DS	LONG	Up to 4 registers	No	Generic	Same as Status SS except: Result is: 0 = intermediate, 1 = off, 2 = on, 3 = bad-state. 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:

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						result for register 1 * 2 <sup>0</sup> + result for register 2 * 2 <sup>1</sup> + result for register 3 * 2 <sup>2</sup> + result for register 4 * 2 <sup>3</sup>
PL Analog Input (L:P:30)	MV/CM	REAL	3 registers	No	CM/PM	<p>First register (300–399 inclusive) indicates that this is an analog input register.</p> <p>Second register is treated as a signed value.</p> <p>Third register can contain a value from –3 to 3 and will be used to scale the second register (R2*10<sup>0</sup>R3).</p> <p>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.</p>
Breaker Rack Status (L:P:230)	EN	LONG	2 to 3 registers	No	Generic	<p>Register 1 = breaker racked in</p> <p>Register 2 = breaker racked out</p> <p>Register 3 = breaker in test (optional)</p> <p>Results:</p> <p>0 = racked in</p> <p>1 = racked out</p> <p>2 = test</p> <p>3 = error</p> <p>4 = in between positions</p>
PL Analog Output (L:P:31)	MV/CM	REAL	2 registers	No	CM/PM	<p>First register (400–499 inclusive) indicates that this is an analog output register.</p> <p>Second register is treated as a signed value.</p> <p>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.</p>
Scaled Register Signed (L:P:32)	MV/CM	REAL	1 or 2 registers	Either (optional)	Generic	<p>For a single register: treated as a signed value from –32,767 to +32,767. (-32768 will result in a NA)</p> <p>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.</p> <p>Values can be scaled using a fixed scale or a scale register.</p> <p>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.</p>



Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
Scaled Register Unsigned (L:P:33)	MV/ CM	REAL	1 to 4 registers	Either (optional)	Generic	For a single register: treated as an unsigned value from 0 to 65,535.  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/ CM	REAL	1 or 2 registers	Either (optional)	Generic	Same as Scaled Register except that a single register with value -32768 is acceptable and will be reported as such.  NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Scaled Register Signed SEPAM A (L:P:35)	MV/ CM	REAL	2 registers	Either (optional)	Generic	Same as Scaled Register except that 0xFFFFFFFF or 0x00007FFF will be NA.  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/ CM	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/ CM	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/ CM	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	PowerSC ADA 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/CM	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/CM	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/CM	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/CM	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/CM	REAL	2 to 4 registers	Either (required)	Generic	Result is: $\text{Avg}(R1 \dots Rn-1) * Rn * 10^{\text{scale}}$ NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Sum with Scale (L:P:45)	MV/CM	REAL	2 registers	Either (required)	CM/PM	Result is: $(R1 * 10^{\text{scale}}) + R2$ NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Sum with Scale Unsigned (L:P:46)	MV/CM	REAL	2 registers	Either (required)	CM/PM	Result is same as above, except unsigned. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
Vector Math (L:P:47)	MV/CM	REAL	2 registers	Either (required)	Generic	Result is: $\sqrt{R1^2 + R2^2} \times \text{scale}$ NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Vector Math IEEE (L:P:48)	MV/CM	REAL	4 registers	Either (required)	Generic	Result is: $\sqrt{([R1 R2]^2 + [R3 R4]^2) \times \text{scale}}$ where [ ] indicates IEEE32 representation NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Multiply Registers 32-bit (L:P:49)	MV/CM	REAL	3 or 4 registers	Either (optional)	Generic	Result is: $[R1R2] * [R3(R4)]$ , meaning Regs 1 and 2 are a 32 bit number. The number is multiplied by Reg 3 (if 16 bit) or Reg 3 and 4 (32 bit number) NOTE: This logic code (and all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
CM4 Power Factor IEEE (L:P:50)	MV/CM	REAL	1 register	No	CM4	Returns the IEEE power factor.
PM8 Power Factor IEEE (L:P:51)	MV/CM	REAL	1 register	No	PM8	Returns the IEEE power factor (converted from IEC mode as necessary). The device may be in IEEE or IEC mode if the device firmware version is 11.6 or higher. If the device firmware version is below 11.6, IEC mode is not supported. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
SP40 Power Factor IEEE (L:P:52)	MV/CM	REAL	1 register	No	SEPAM 40	Returns the IEEE power factor (converted from IEC mode). NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
ML Power Factor IEEE	MV/CM	REAL	2 registers	No	ML	Returns the IEEE power factor (converted from IEC mode as necessary).

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
(L:P:53)						The second input register must be the associated Reactive Power for the Power Factor requested. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Generic Power Factor (L:P:54)	MV/CM	REAL	2 registers	No	Generic	$R2/\sqrt{R2^2 + R1^2}$ where: R2 = real power R1 = reactive power NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Generic Power Factor - IEEE32 variation (L:P:55)	MV/CM	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/CM	REAL	1 register	No	SEPAM 2000	Returns the IEEE power factor (converted from IEC mode). NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Scaled Register Signed 64-bit (L:P:57)	MV/CM	REAL	4 registers	Either (optional)	Generic	Reads a 64-bit signed integer and returns a REAL value. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
Power Factor IEEE (L:P:58)	MV/CM	REAL	2 registers	No	Generic	Takes a 4 quadrant power factor (IEEE32 real) and returns an IEEE power factor
IEEE 64-bit double (L:P:59)	MV/CM	REAL	4 registers	No	Generic	Uses the IEEE standard for floating-point arithmetic (IEEE 754); returns the value as 32-bit REAL. NOTE: This logic code (as with all REAL logic codes) has an accuracy of seven digits. Anything longer than seven digits should not be considered accurate.
IEEE 64-bit double (L:P:60)	MV/CM	STRING	4 registers	No	Generic	Uses the IEEE standard for floating-point arithmetic (IEEE 754); returns the value as 64-bit STRING.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
						NOTE: This logic code (as with all REAL logic codes) has an accuracy of 15 digits. Anything longer than 15 digits should not be considered accurate.
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:10 1)	SS	LONG	1 register	No	Generic	If you input 1 to this tag it will write the <b>MASK</b> value to the register.
Status Write Register AND (L:P:10 2)	SS	LONG	1 register	No	Generic	If you input 1 to this tag it will read the register and <b>AND</b> the <b>MASK</b> 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:10 3)	SS	LONG	1 register	No	Generic	If you input 1 to this tag it will read the register and <b>OR</b> the <b>MASK</b> 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:11 0)	MV/CM	REAL	1 register	Either	Generic	This will take the input value read in and divide out the scale factor and the conversion factor. It will then round to the nearest whole number and if it is a value from 0 to 65535 it will put this value in the register.  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:11 1)	MV/CM	REAL	1 register	Either	Generic	This will take the input value read in and divide out the scale factor and the conversion factor. It will then round to the nearest whole number and convert the signed value to an unsigned value from 0 to 65535. It will put this value in the register.  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	MV/CM	LONG	1 register	No	Generic	You can write any value from 0 to 65535 and read an unsigned value from the same register.

Logic Code	IEC Type	PowerSC ADA Expert Data Type	Register Definition	Scaled Required? (register or fixed)	Device Specific?	Notes
(L:P:120)						
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.

## Add a Project Using the Project Setup Tool

There are two methods for adding a project:

You can use the PowerSCADA Studio, (see *The Design Time Environment*), which is a more manual process. Use PowerSCADA Studio to ensure that you have covered everything, and for further modification of your project.

Or, you use this tool to rapidly deploy a project. Using Project Setup, you can do the following:

- create and name a project
- select screen resolution and determine whether you will use normal or high contrast
- specify primary and secondary server connections
- specify the Advanced Reports and Dashboards connection
- add users and Windows authentication
- add devices
- add default pages to your project
- add menus to display pages in runtime
- choose the landing page for each monitor in a multi-monitor project

If you are using optional components (see below), you need to install them separately.

After you create the project and its features, you can edit other settings, such as devices in the CSV file, and use it to update your project.

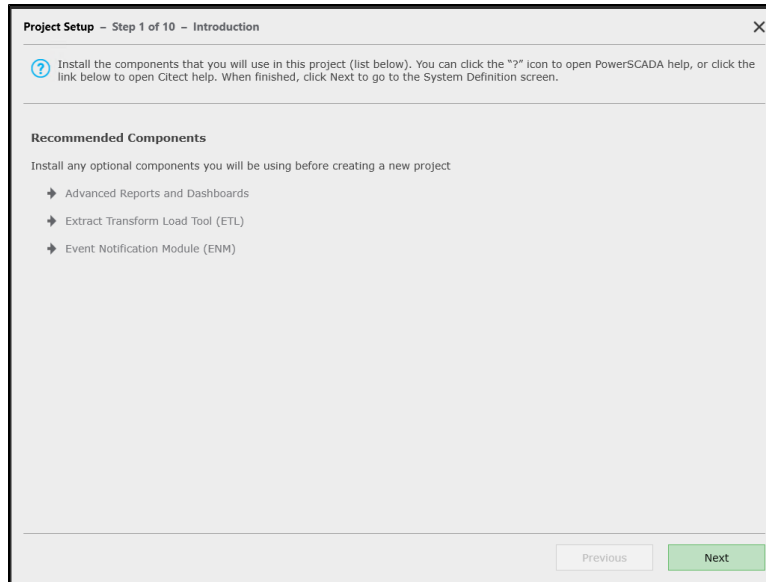
You save time when you use Project Setup, as you are able to change a variety of parameters. When you do not use Project Setup, you then must navigate to a variety of locations to change or set up these parameters. For a list of parameters that you can set in Project Setup, see:

[Parameters Changed In Project Setup on page 106](#)

For help on any of the Project Setup pages, you can:

- click "?" to the left of the top line to view the entire PowerSCADA Expert help file
- hover your mouse over fields to read tooltips

To launch Project Setup, launch the PowerSCADA Studio. From the Projects tab, click Add > Create and Configure Project. You see the Introduction page.

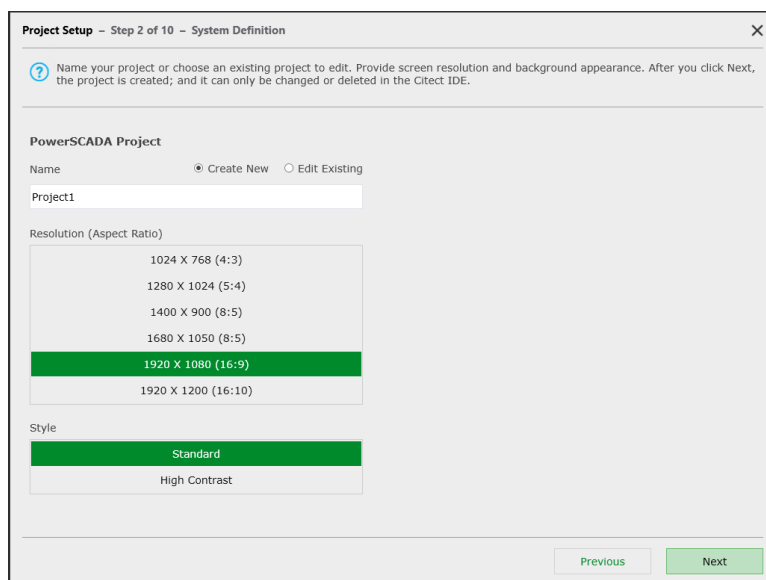


This page lists optional components that you may want to install. The install files are on the root of the PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards installation media.

- PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. Use this component to view advanced reports and dashboards from Power Monitoring Expert. Install this component from the PowerSCADA Expert installation media.
- Extract Transform Load (ETL) module. Use this component to extract reporting information from PowerSCADA Expert and transfer it to Power Monitoring Expert, for use in reports).
- Event Notification Module (ENM). Use ENM to set up email and text messaging for project alarms.

Click Next to display the System Definition screen.

## System Definition





On this screen, you determine whether you are creating a new project or editing an existing one, and the "view" of the project.

1. Click either Create New or Edit Existing.
2. If you click Create New, type a project name. If you click Edit Existing, choose a project from the list that displays. Use only alphanumeric characters and underscores.
3. In the Resolution box, choose the screen resolution that you want for the graphics pages in this project. This should match the resolution of the monitor that will display graphics pages.
4. In the Style box, choose whether you want Standard or High Contrast. Standard uses a white background. High contrast uses a black background, which makes it easier to view graphics pages.
5. Click Next to display the Servers page.

**NOTE:** After you click Next, the project is created. You cannot change or delete it here. To make changes or to delete it, you must use the PowerSCADA Studio.

You can also set high contrast via the parameters in the PowerSCADA Studio. Open your project in the PowerSCADA Studio, then click Settings > Parameters. The parameter name is **IsHighContrast**. 0 = normal; 1 = high contrast.

## Servers

Project Setup - Step 3 of 10 - Servers and Web Client

Enter the address information for the servers in this project. If two or more servers are in the startup project file, you can only add an address and user account for Advanced Reports and Dashboards.

**Topology**

Server Name or IP Address  
127.0.0.1

Redundant System  
Standby Server Name or IP Address

Advanced Reports and Dashboards  
Advanced Reports Server Name or IP Address

User Name Password Confirm Password

Previous Next

Use this screen to define the server information for your primary server and the server for Advanced Reports and Dashboards.

The Project Setup detects the number of servers that are in your starter project. If you only have one server (for example, using the loopback IP address), you see all of the fields in the screen above. If you are using a project that has two or more servers identified, you only see the bottom section, Advanced Reports and Dashboards.

1. Server Name or IP Address: Type the Server Name or IP Address for the project's primary server, or select it from the drop-down list.
2. Redundant System: If this is a redundant system, check Redundant System. Checking this box enables the Standby Server Name or IP Address prompt.
3. Standby Server Address: Type the Server Name or IP Address of the standby server, or select it from the drop-down list.

4. **Advanced Reports and Dashboards:** If you have installed the Advanced Reports and Dashboards component (WebReach is also assumed to be on this server), check this box, which will enable the Advanced Reports prompts.
5. **Advanced Reports Server Name or IP Address:** Type the Server Name or IP Address of the Advanced Reports server, or select it from the drop-down list.
6. **User Name/Password/Confirm Password:** These fields are only enabled after you check Advanced Reports and Dashboards. Type the user name and password used for the Advanced Reports and Dashboards server. (Note that you must enter a user name and password if you check the Advanced Reports and Dashboards box.)

Retype the password in the Confirm Password field.

7. Click Next to view the Users screen.

## Users

**Project Setup - Step 4 of 10 - Users**

Add the PowerSCADA user account information for each user who will access this project. Each user must be assigned to a role. Each role can also be a member of a pre-established Windows group.

**PowerSCADA Users**

[Add User](#)

User Name	Role	Password	Confirm Password	Full Name (optional)
aol	Role0	*****	*****	aol

[Delete Selected](#)

**Windows Authentication - Active Directory (optional)**

Role	Windows Group
Controller	
Operator	
Role0	

[Previous](#) [Next](#)

On this screen, you define PowerSCADA Expert users, passwords, and roles. Add the PowerSCADA user information for each user who will access the runtime pages in this project.

Each user must be assigned to a role. You can also associate each role with a pre-established Windows group. For more information about users and roles, see [Add and Modify User Accounts on page 237](#).

1. To begin adding an account, click Add User; a blank row displays in the user list. If you are editing a user, click the user name.
2. Click the Role column for the user, and choose the appropriate role:

If you use the default startup project, you see these default roles:

- Controller
- Operator
- Role0
- Supervisor
- User

If you have edited the roles in the default startup project of the PowerSCADA Studio, this list will match the edited startup project that you are

using.

3. To delete any user that you have added, highlight the user line and click Delete Selected.
4. Type and confirm the password to be used by this user.
5. If desired, add a full name for this user. This simply provides a place for a more descriptive name; it is not used to log on to the system.
6. In the Windows Authentication box, you can optionally assign a role to a Windows group.

This provides central management of users through Windows. It also means that Windows users who are in the specified Windows group will have the PowerSCADA Studio privileges that are assigned to this role.

See "Use Windows Integrated Users" in [Add and Modify User Accounts on page 237](#) for more information.

7. Click Next to display the Menus and Display Pages.

## Menus and Display Pages

**HMI Menu**

- Home
- Graphics
- Single Lines
- Alarms / Events
- Analysis (Process Analyst, Instant Trend, Waveform, Tag Viewer)
- Advanced Reporting
- Dashboards

**Monitors**

Total Monitors: 1 [ - ] [ + ]

**Runtime Landing Page**

Monitor 1: Elevations	Monitor 2: Elevations
Monitor 3: Elevations	Monitor 4: Elevations
Monitor 5: Elevations	Monitor 6: Elevations
Monitor 7: Elevations	Monitor 8: Elevations

Previous Next

Use this screen to accomplish two tasks:

- determine the top-level menu items that will display in the runtime graphics pages
- determine the page that will display on any of the monitors that you have in a multi-monitor installation

### HMI Menu

Design the Level 1 (top line) menu structure for graphics pages: check the menu tabs that you want to display on the graphics page. You can add more detail, such as secondary levels, in the Menu Configuration page (PowerSCADA Studio > Visualization tab > Menu Configuration).

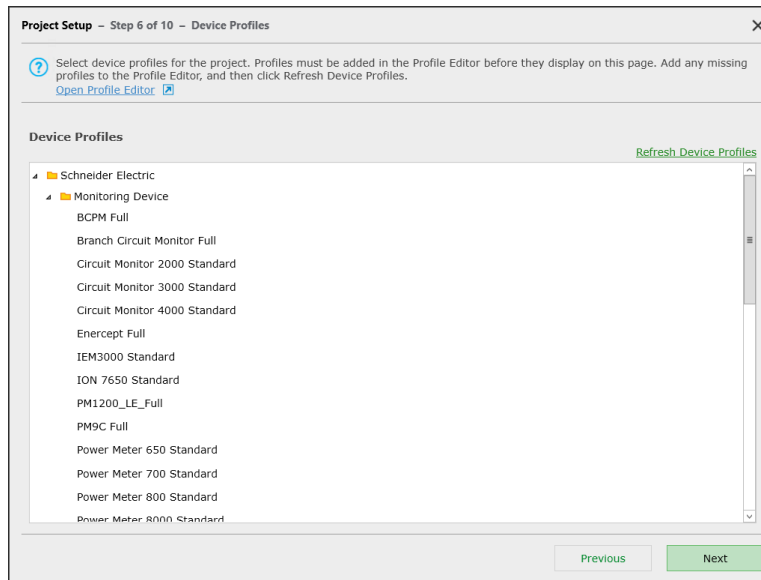
### Monitors

If you have multiple monitors in your system, type the number (up to 8) of monitors in the Total Monitors box. You can also click the plus and minus buttons to increase or reduce the number. The Runtime Landing Page prompts corresponding to the number of Total Monitors will become enabled.

For each monitor, click the drop-down menu to display available pages. Choose the landing page you want to see when this monitor views PowerSCADA Expert.

When you are finished, click Next to display the Summary screen.

## Device Profiles

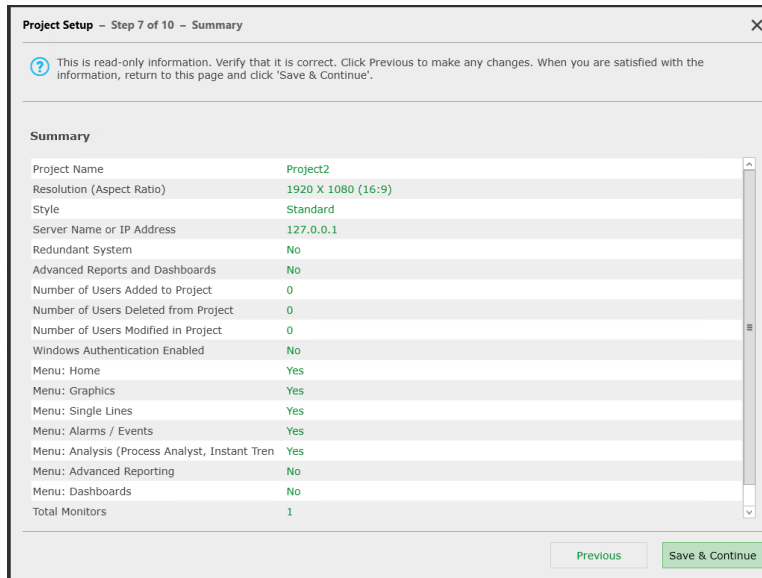


This screen displays device profiles for the project. Profiles will display here only if they exist in the Profile Editor and have been exported to this project.

If a profile is missing from this list, click **Open Profile Editor** to launch the Profile Editor. Create the profile if it doesn't already exist and/or add it to the project there and export the project. Return to this page and click **Refresh Device Profiles** to update the list. Likewise, if you need to remove a profile, launch the Profile Editor, remove the profile from the project, export the project, and then click Refresh Device Profiles.

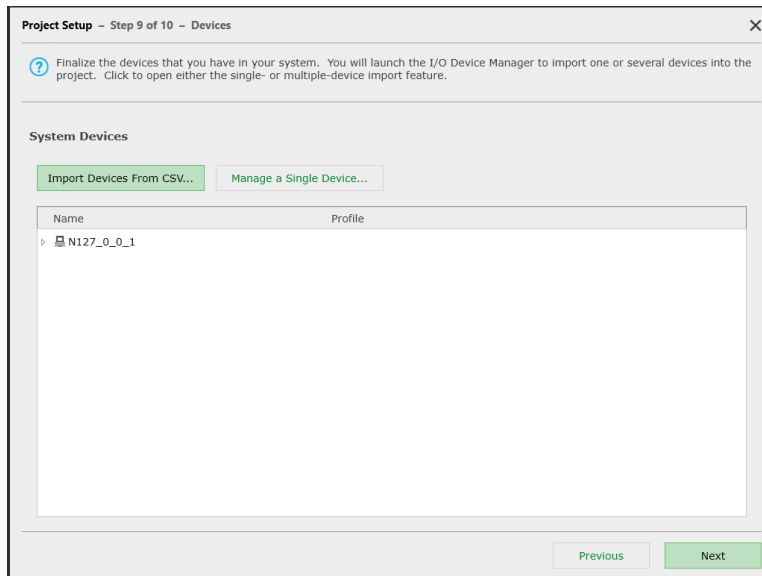
When you see the profiles that you want in your project, click Next to view the Devices page.

## Summary



This is read-only information. Verify that it is correct for your system. If you need to change something, click Previous to return to that screen. When you are satisfied with the information, click Save and Continue to display the Device Profiles page.

## Devices



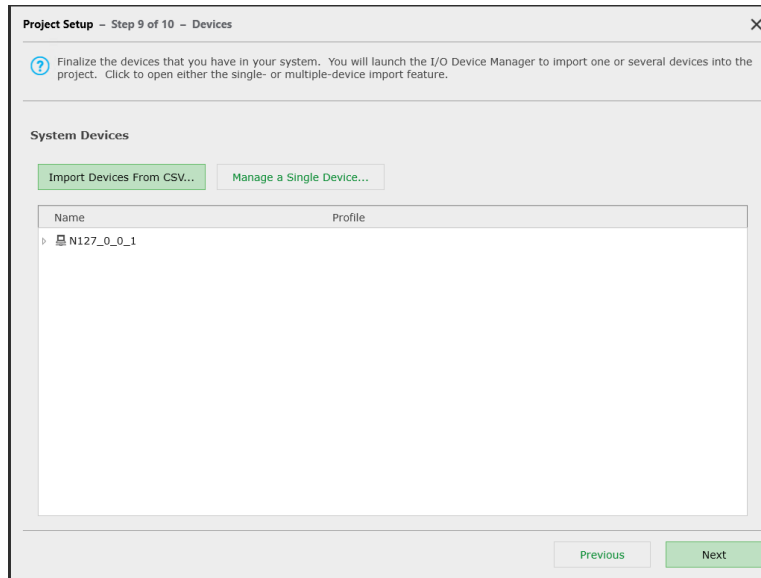
Finalize the devices that you have in your system. Use the IO Device Manager to import one or several devices into the project.

To import multiple devices from a CSV file into your project, click Import Devices From CSV. For detailed instructions, see [Use a CSV File to Define Multiple Devices on page 132](#)

To import a single file, click Manage a Single Device. For more information, see [Define One I/O Device in a Project on page 118](#).

Click Next to view the Finish page.

## Finish Page



Before you start work on this page, you can click "Open System Development Guide" to find instructions on creating a project from beginning to end.

On this screen, compile the project. After it is compiled, it will open in the runtime environment. In runtime, verify correct device communication using the Tag Viewer screen or by viewing any Communication Failure Alarms.

In the Project Development Tools section, you can:

- launch the PowerSCADA Studio to make a variety of changes to the project
- launch the Graphics Builder page to make changes/additions to the graphics page you are creating
- launch the Application Configuration Utility to edit or set up many project features (see *Application Configuration Utility* for instructions)

Many of the settings made by Project Setup are also included in the Parameters file (PowerSCADA Studio > Settings > Parameters). You can also change these parameters in that file.

When you are finished, click Finish to leave Project Setup.

## Parameters Changed In Project Setup

In the Project Setup wizard, you can easily set up a variety of project information. You can edit many project parameters. Because the parameters are in a variety of places, you will save time when you use Project Setup to quickly edit them. The parameters below are organized according to the Project Setup page that lets you edit them.

### System Definition screen

Project Setup Setting	Section	Parameter Name
Resolution	MultiMonitors	Resolution
Style	MultiMonitors	IsHighContrast

### Servers

Project Setup Setting	Section	Parameter Name
Advanced Reports Server	Applications	Hostname

### Menus in Project Setup

For each page selected in Project Setup (Step 5), the menu configuration items are added.

Project Setup Setting	Section	Parameter Name
Monitor Count	MultiMonitors	Monitors
Monitor 1 Landing Page	MultiMonitors	StartupPage1
Monitor 2 Landing Page	MultiMonitors	StartupPage2
Monitor 3 Landing Page	MultiMonitors	StartupPag3
Monitor 4 Landing Page	MultiMonitors	StartupPage4
Monitor 5 Landing Page	MultiMonitors	StartupPage5
Monitor 6 Landing Page	MultiMonitors	StartupPage6
Monitor 7 Landing Page	MultiMonitors	StartupPage7
Monitor 8 Landing Page	MultiMonitors	StartupPage8

In addition to parameters, you can do the following:

### Servers, Network Addresses, and Computers

Project Setup Location	Item
Step 3, Servers	Add I/O, Alarm, Trend, and Report Servers, primary and redundant NOTE: Clusters are also added here.
Step 3, Servers	Add network addresses, primary and redundant
Step 5, Display: Menus and Display Pages	Create HMI menus: setup for graphics pages
Step 5, Display: Menus and Display Pages	Determine runtime landing pages at various monitors used in the project
Step 7: Device Profiles	Choose device profiles
Step 8: Devices	Add I/O devices; including equipment, ports, boards, I/O devices, variable tags, alarm tags, trend tags
Step 9: Finish	Compile and run the completed project

# 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 108](#)

[The PowerSCADA Expert Pages on page 109](#)

[Create a Project on page 109](#)

[Use PowerSCADA Expert One-Lines](#)

[One-Line Flowchart on page 134](#)

[What Are PowerSCADA Expert One-Lines? on page 135](#)

[One-Line Device \(zOL\) on page 136](#)

[One-Line Colors on page 137](#)

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 137](#)

[Start and Stop One-Lines on page 139](#)

[Work with Genies on page 139](#)

[Work in the Configuration Utility on page 143](#)

[Work with Alarms on page 154](#)

[Alarms Overview on page 154](#)

[Add Setpoints and Delays on page 155](#)

[Set Up an Alarm Based on an Enumeration on page 155](#)

[Change an Alarm Severity on page 156](#)

[Enable Waveforms for Onboard Alarms on page 156](#)

[Set Parameters for Event Log Length and Historical Logging of Events on page 156](#)

[Add an Onboard Alarm Tag on page 157](#)

[Set Up Audible Alarms on page 157](#)

[Restore and Back Up Projects on page 158](#)

[Restore a Project on page 158](#)

[PowerSCADA Expert Backup on page 158](#)

[Use Cicode to Customize a Project on page 159](#)

[Work in the Configuration Utility on page 143](#)

[Work in the Configuration Utility on page 143](#)

[Trend Tag Scan Intervals on page 163](#)

[Disk Storage Calculation for Trends on page 164](#)

## 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 I/O Device Manager 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 114 for more information about changing the appearance of the graphics page.)

3. Add genies to the page to create the one-line drawing.
4. Compile and run the project. Note any errors and warnings; correct all errors. Review warnings for problems such as missing tags.
5. Run the project to view the graphics page in the runtime environment.

## The PowerSCADA Expert Pages

When you launch the product (v8.2), these windows are opened:

- **PowerSCADA Studio:** 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.
- **Graphics Builder:** Use this application to build graphic display pages for a project. A standard PowerSCADA Expert library of genies is provided. The genies contain logic such as control outputs and rack-in/rack-out. Once assembled and opened in the runtime environment, they provide an interactive view of an installation's power system.

## Create a Project

The information in this section is based on the Normal template from the PLS\_Style\_1 library. The Normal template includes all of the tabs and buttons that are described in the topics here. If you want to edit selections on the tabs and buttons, use the Menu Configuration tool. For instructions on editing templates with the Menu Configuration tool, see [Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\)](#) on page 114.

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 in PowerSCADA Studio, 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 109](#)

[Add a New Graphics Page on page 110](#)

[Set a New Page as the Project Startup Page on page 111](#)

[Create a One-Line on a Graphics Page on page 111](#)

[Use Menu Configuration to Edit Pagemenu.dbf \(Change the Graphics Page Appearance\) on page 114](#)

[Create New Genies on page 115](#)

[Delete Information from PowerSCADA Expert on page 117](#)

## Reuse Projects Created in the Profile Editor

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

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



3. Copy the file (Equipment.profiles) that displays. If you need to use this file to another computer, you can move it to a portable drive.
4. On the server computer, paste Equipment.profiles in this location, where:  
[Drive Letter]: is the drive on which you installed the server  
the Application Data and Program Data folders are not hidden (set the folder view for "view hidden folders")  
[Project] is the name of the project you are creating; you must have already added this project to PowerSCADA Expert (see Before You Export, above).
5. Be sure you have created the files described in [Before You Begin on page 110](#).

## Before You Begin

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

- use the Profile Editor to add all of the device types, device profiles, and projects
- from PowerSCADA Studio, create a project; add clusters, network addresses, and servers
- export devices from the Profile Editor
- add devices into the PowerSCADA Expert project, using the I/O Device Manager

## 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 PowerSCADA Studio Projects screen (see steps below). Then add it to the Profile Editor. See [Add, Edit, or Delete a Project on page 70](#) for instructions.

1. From the PowerSCADA Studio page, 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 114](#) for instructions on making these changes.
5. Save the graphics page, giving it the desired name and selecting the project that you have just added.

## Change the Background Color of Pages

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

1. Create a new project, using "high contrast" as the starter project.

This project will already include all of the pages that you will use. There will also be a new parameter, called *PageColorExt*. This parameter includes the extension that allows you to color backgrounds.

2. To designate background color for each page, pick the color in the Graphics Editor.

You can also change the genie font colors on your project pages. To do this see [Change the Genie Color in Project Pages on page 111](#).

## Change the Genie Color in Project Pages

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

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

**NOTE:** If you use the "high contrast" starter project to create your project, this parameter is automatically added, as are all of the pages that you will use. For instructions on changing the background color of your project pages, see [Change the Background Color of Pages on page 110](#).

## 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.

**NOTE:** You can also set a separate startup page (landing page) for each monitor in a multi-monitor system. For instructions, see [Menus and Display Pages on page 103](#).

Setting\_ To set the startup page on the main server:

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

## Create a One-Line on a Graphics Page

You will build a one-line by adding genies to the new page. For more information about genies, see **Genies and Super Genies** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

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

1. Click the "genie" icon on the toolbar:



2. From the Paste Genie screen, choose a library (to ensure that all PowerSCADA Expert features are available, choose a library that begins with "pls"). If you want to edit a genie or create a new one, see [Create New Genies](#) for instructions.

3. From the available icons in the selected library, select a genie; click OK to paste the genie on the graphics page.

4. A popup window displays for you to enter genie properties. Enter the requested information for the appropriate object:

*Configure a Busbar* on page 140

*Configure a Circuit Breaker or Switch* on page 141

*Configure a Meter* on page 141


*Configure a Source* on page 142

*Configure a Transformer* on page 142

**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 Citect Engineering Environment. Click

Options () on the left hand side. From the Options dialog box, de-select “Prompt on tag not exist.” Click OK.

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. Compile the project: from the left hand side of PowerSCADA Studio, click 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 Projects > Pack to clean up the files.

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

12. After all errors are corrected, run the project (click ).

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 114 for help.

## Enable Lockout/Tagout



### DANGER

#### HAZARD OF ELECTRIC SHOCK, EXPLOSION, OR ARC FLASH

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

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

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

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

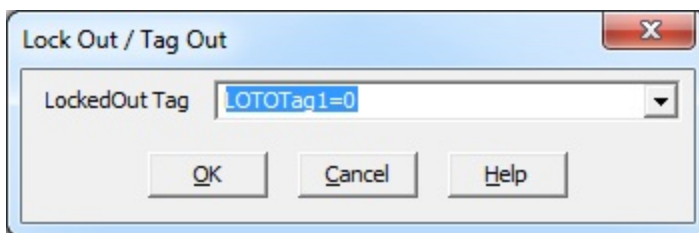
This is a read-only feature; but it does not prevent controls to the device or area. This feature is not available in PLS\_Example.

Follow these steps to enable the icon for a device:

1. From the pls\_gen\_eq\_2 genie library, add the lockout/tagout genie to the graphics page. Position it beside the equipment that is being monitored.
2. From Variable Tags, choose the device tag (or create a custom tag) that you want, and associate it with the device that will read the lockout/tagout status.
3. Define what indicates the status (for example, *door status open*, 0 = false and 1 = true).
4. Edit the lockout/tagout genie to read the tag on the device. Make the lockout/tagout genie visible when the device tag indicates that the device is in lockout/tagout status; and make the icon hidden when the device is not in lockout/tagout. (By default, lockout/tagout status is hidden when the tag property is "NOT [variable tag].")

To continue the example, if the tag is reading *door status open* and the value = 0, then the door is open and in lockout/tagout; if the value = 1, the door is closed and not in lockout/tagout.

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



## Use Menu Configuration to Edit Pagemenu.dbf (Change the Graphics Page Appearance)

The Menu Configuration form (PowerSCADA Studio > Visualization tab > Menu Configuration) 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 Menu Configuration page for the PLS\_Example project (see the table below for descriptions of the columns).

Row	Level 1	Level 2	Level 3	Level 4	Menu Command	Comme	Order	Symbol	Page	Project
1	Home				PLSNavPageHome()			pls_icons.greer		PLS_Example
2	Single Lines							pls_icons.greer		PLS_Example
3	Single Lines	Overview						PLS_Icons.Over		PLS_Example
4	Single Lines	Overview	ANSI Style		PLSPageDisplay("OVER\			PLS_Icons.Over		PLS_Example
5	Single Lines	Overview	IEC Style		PLSPageDisplay("OVER\			PLS_Icons.Over		PLS_Example
6	Single Lines	12.47 kV Subst			PLSPageDisplay("SLD_3			PLS_Icons.sub		PLS_Example
7	Single Lines	4.16 kV Substa			PLSPageDisplay("SLD_6			PLS_Icons.sub		PLS_Example
8	Single Lines	480 V Substati			PLSPageDisplay("SLD_4			PLS_Icons.Over		PLS_Example
9	Single Lines	480 V Substati			PLSPageDisplay("SLD_4			PLS_Icons.Over		PLS_Example
10	Single Lines	480 V Substati			PLSPageDisplay("SLD_4			PLS_Icons.Over		PLS_Example
11	Alarms / Event							pls_icons.greer		PLS_Example
12	Alarms / Event	Event Log			PLSDspShowAlarm(15)			PLS_Icons.Over		PLS_Example
13	Alarms / Event	Alarm Log			PLSDspShowAlarm(0)			PLS_Icons.alar		PLS_Example

Menu Item	Description
Levels 1 through 4	These items establish the menu levels that will display. For example, you might use "Single Lines" for level 1, followed by the substation for level 2, and the graphic name for level 3. (Each line: 256 characters maximum)
Menu 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.
Order	The relative position within the final graphics page. If you leave this field blank, the default value 0 is used. (64 characters maximum)
Symbol	Displays a defined image along with the description for that level.

Menu Item	Description
	<p>Images must already be defined in the project/include project. They are specified in the format &lt;library name&gt;, &lt;symbol name&gt;. For example, in PLS_Example, the symbol used for the level 2 of Single Lines is Substation3, entered as <b>PLS_Icons.Substation3</b>.</p> <p>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.</p>
Page	The page on which this entry will display. If this is left blank, the entry will display on every page.
Comment	You can use up to 128 characters to add a comment (will not display on screen).

To make it easier to create the menu for a project, you can copy entire lines from PLS\_Example or another project into the new project.

## 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 143](#).

**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 143](#).

The easiest way to create a new genie is to make a copy of a similar genie from the standard library and edit it in your project. We recommend, when adding layers, that you keep the dimensions of the new layers the same as the original. For a list of all of the standard PowerSCADA Expert genies, including all of the smaller parts that you could use to create genies, see [Appendix 1: Default Genie Library on page 239](#).

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 the Graphics Builder, click File > Open. Click the Genie tab, then select the library and the template genie. Click OK.
2. Save the genie with a new name in your local project.
3. To separate the genie into layers, click the genie and drag a layer to the bottom right.
4. Repeat the process to pull all of the layers apart.  
Each layer is a sub-genie that controls a different aspect of the overall genie.
5. Make the changes that you wish to the genie.  
**NOTE:** Be careful to maintain the same dimensions for any new layers that you create.
6. To re-assemble the genie, draw a marquis around all of the parts; then click Arrange > Align, and choose Top and Left. Click OK.

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

You can rotate any of the genies.

### To create a custom symbol for a custom genie:

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

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

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

See the default symbols in the PowerSCADA Expert libraries that begin with PLS\_.

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

## Set Up IEC 61850 Advanced Control

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

See [Perform IEC 61850 Advanced Control on page 185](#) for information on using these advanced controls.

### WARNING

#### UNINTENDED EQUIPMENT OPERATION

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

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

#### Enable the Advanced Control

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

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

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

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

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



## Delete Information from PowerSCADA Expert

If you find you need to delete any data that you have entered (clusters, servers, genies, and so on), follow the normal procedure to delete the data, then use the Pack command to completely delete it. To do this, launch the PowerSCADA Studio; from the Projects tab, click Pack.

## Manage I/O Devices in the Project

The I/O Device Manager tool provides a quick way to add one or many devices to a PowerSCADA Expert project. Instead of adding device and device manager Rather than having to "build" devices one tag at a time in the PowerSCADA Studio, use the I/O Device Manager to create or many devices, and to update or remove devices.

To begin, launch the I/O Device Manager from Start menu (Start > Schneider Electric > IO Device Manager).

Click *Import Devices from CSV* to go to a wizard that walks you through managing devices that are in a CSV file. Or click *Manage a Single Device* to go to a wizard that walks you through creating, removing, or updating a single device.

Throughout the I/O Device Manager 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.

Click a link for what you want to do.

- [Define One I/O Device in a Project on page 118](#)
- [Remove a Single I/O Device From a Project on page 127](#)
- [Use a CSV File to Define Multiple Devices on page 132](#)
- [Update One or All Devices in Project on page 132](#)

The following links provide additional information that you need to know.

[When You Set Up More than two I/O Servers per Cluster on page 22](#)

[Port Names on page 24](#)

## Before You Use the I/O Device Manager

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 Studio.

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

**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 111](#).

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.

## Define One I/O Device in a Project

Use the wizard in the I/O Device Manager to add one device at a time. Launch the wizard from the Start menu (Start > Schneider Electric > IO Device Manager). From the I/O Device Manager subtab, click *Manage a Single Device* to open the wizard..

Throughout the I/O device wizard, there are fields that will only accept a valid entry. They are marked with a red exclamation point (!). The exclamation point remains there until you enter a response that is of the correct length or includes only the acceptable characters. The exclamation point disappears when you have entered a valid response.

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

[Add a TCP Device to a Project on page 118](#)

[Add a Serial Device to a Project on page 120](#)

[Add a DNP3\\_TCP Device on page 123](#)

[Add an IEC 61850 Device to a Project on page 125](#)

### 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*).

Then, make sure that you have set up each of these clusters in the I/O Device Manager:

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 I/O Device Manager, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see [When You Set Up More than two I/O Servers per Cluster on page 22](#)).
5. At the Ready to perform action screen, click Next.

If you have more than one cluster to add, repeat steps 3 through 5 for each cluster.

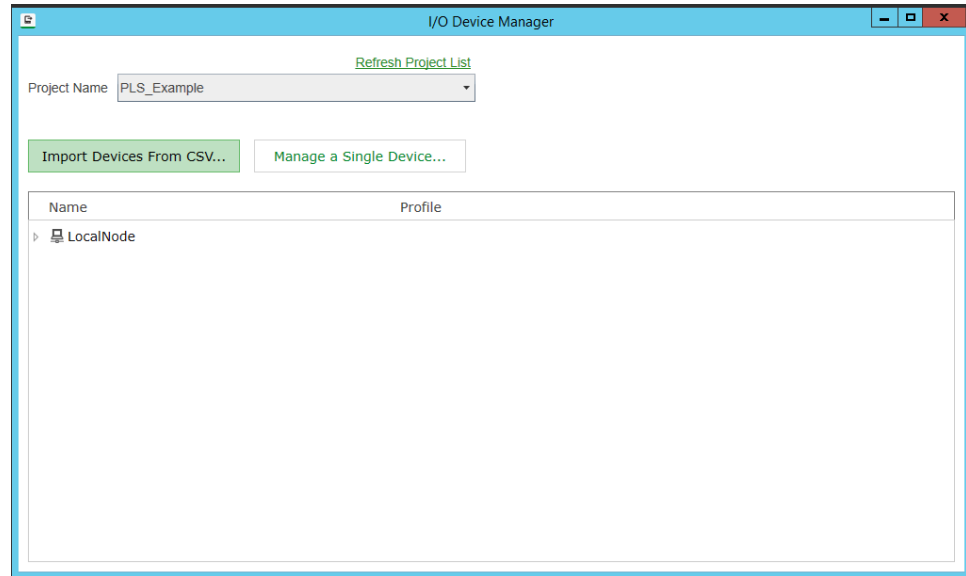
When you are finished adding clusters and I/O servers, you return to the I/O Device Manager welcome screen.

### Add the TCP Device

Follow these steps to add TCP devices:

1. From the Start menu, run the I/O Device Manager (Schneider Electric > IO Device Manager).

The I/O Device Manager screen displays:



2. Click *Manage a Single Device*.
3. From the new I/O Device Manager screen, click *Create one I/O Device*, then click *Next*.
4. At the Choose profile screen, select the first device profile that you want to use to add a device to the project. Click *Next*.

**NOTE:** To ensure that the Alarm Log displays properly with the PM5000 series of devices, use the correct PM5000S or PM5000S1 driver for devices:

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

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

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

- PM5350 with FW prior to version 3.00

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

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

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

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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. If you choose to add an optional sub-profile: 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 24](#).)
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 I/O Device Manager closes. If you checked the Add/remove ... box, the Welcome screen displays again.
12. From any of the three PowerSCADA Expert screens, compile the project.
13. When all errors are corrected, click 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](#)).

Then, make sure that you have set up each of these clusters in the I/O Device Manager:

**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):

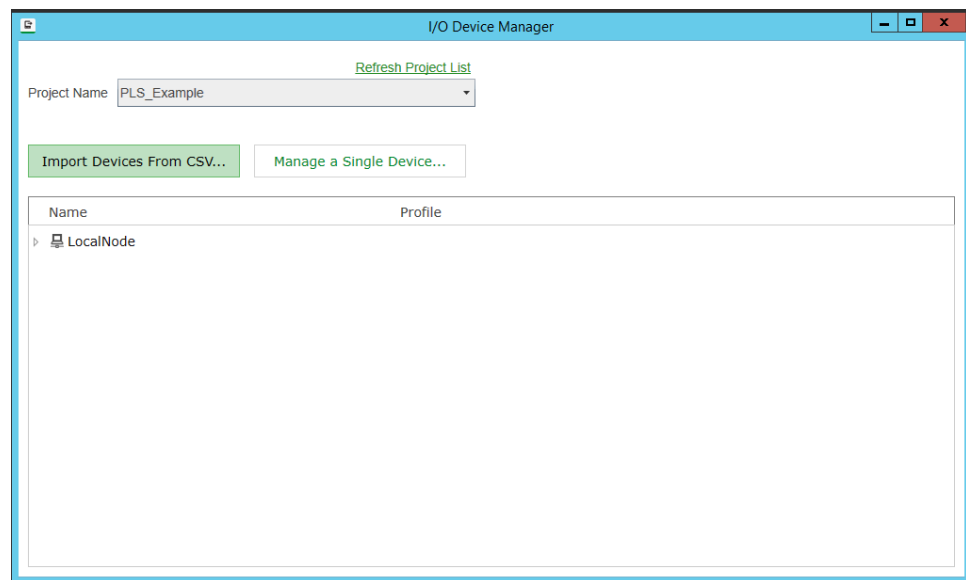
1. From the I/O Device Manager, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see [When You Set Up More than two I/O Servers per Cluster on page 22](#)).
5. At the Ready to perform action screen, click Next.  
If you have more than one cluster to add, repeat steps 3 through 5 for each cluster.  
When you are finished adding clusters and I/O servers, you return to the I/O Device Manager welcome screen.

## Adding the Serial Device

Follow these steps to add serial devices.

1. From the Start menu, run the I/O Device Manager (Schneider Electric > IO Device Manager).

The I/O Device Manager screen displays:



2. Click *Manage a Single Device*.
3. From the new I/O Device Manager screen, click *Create one I/O Device*, then click Next.
4. At the Choose profile screen, select the first device profile that you want to use to add a device to the project. Click Next.

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

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

- PM51XX
- PM53XX
- PM55XX

- PM5350PB
- PM5350IB with FW version 3.00 and higher

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

- PM5350 with FW prior to version 3.00

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

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

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

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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 add information for 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 all of the information for each server (com port, baud rate, and so on). If you check the "Same as Primary" box for standby, you will use the same addresses for the primary and standby. Click Next.
9. At the Port Settings screen, you can rename each of the ports. (See more information about naming ports in [Port Names on page 24](#).)
10. Repeat steps 6 through 8 for the remaining sub-profiles.
11. 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.

12. 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 I/O Device Manager closes.

13. From any of the three screens, compile the project.
14. When all errors are corrected, click 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*).

Then, make sure that you have set up each of these clusters in the I/O Device Manager:

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 I/O Device Manager, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see [When You Set Up More than two I/O Servers per Cluster on page 22](#)).
5. At the Ready to perform action screen, click Next.

If you have more than one cluster to add, repeat steps 3 through 5 for each cluster.

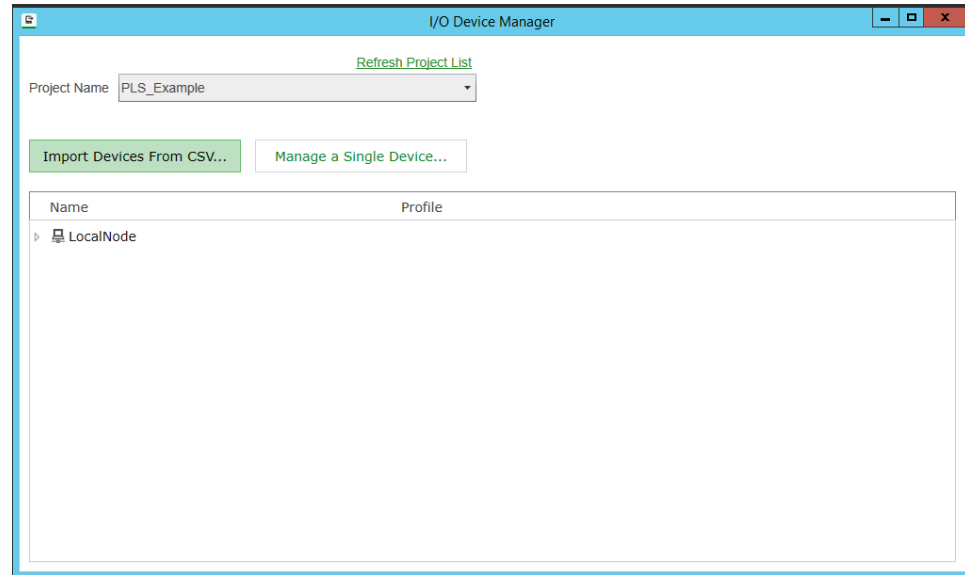
When you are finished adding clusters and I/O servers, you return to the I/O Device Manager welcome screen.

### **Add the DNP3\_TCP Device**

Follow these steps to add TCP devices:

1. From the Start menu, run the I/O Device Manager (Schneider Electric > IO Device Manager).

The I/O Device Manager screen displays:



2. Click *Manage a Single Device*.
3. From the new I/O Device Manager screen, click *Create one I/O Device*, 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 8.2 with Advanced Reporting and Dashboards\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 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.
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 24.](#))
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 I/O Device Manager closes. If you checked the Add/remove ... box, the Welcome screen displays again.
13. From any of the three screens, compile the project.
14. When all errors are corrected, click Run () to view the runtime environment.

### Add an IEC 61850 Device to a Project

Use this procedure when you want to add an IEC 61850 device to a project. Note that these instructions assume that you have two I/O servers, and that you will be renaming ports.

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

Before you begin, make sure that you have added at least one cluster and the appropriate servers for this project (see [Add the Cluster, Network Address, and Servers](#)).

Then, make sure that you have set up each of these clusters in the I/O Device Manager:

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 I/O Device Manager, under System Devices, click Cluster Setup. Click Next.
2. At the Enter Instance Information screen, a cluster name displays. Click Next.
3. If there are multiple clusters, the Select cluster screen displays. Choose the cluster you want to set up. Click Next.
4. If there are multiple I/O servers in the cluster, the Select I/O servers screen displays. Check *Supports Redundancy* and select the I/O servers to which you want to add

the device. Click Next. If there are more than two IO servers per cluster, you must set them up manually (see [When You Set Up More than two I/O Servers per Cluster on page 22](#)).

- At the Ready to perform action screen, click Next.

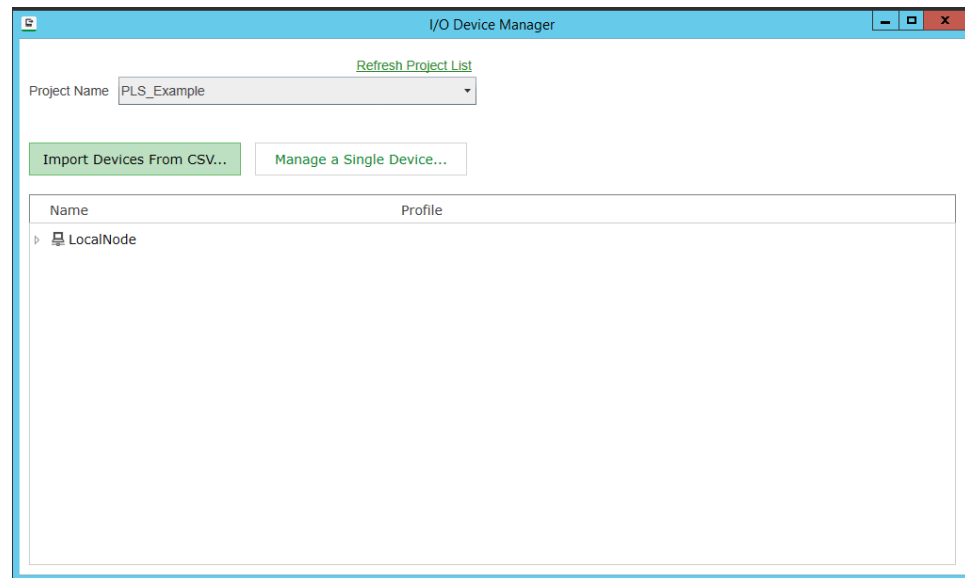
If you have more than one cluster to add, repeat steps 3 through 5 for each cluster.

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

## Add the IEC 61850 Device

- From the Start menu, run the I/O Device Manager (Schneider Electric > IO Device Manager).

The I/O Device Manager screen displays:



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

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

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

the copy you typed in the comment field is added to the default language, named English.DBF. After the project is compiled, this file is located in Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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.

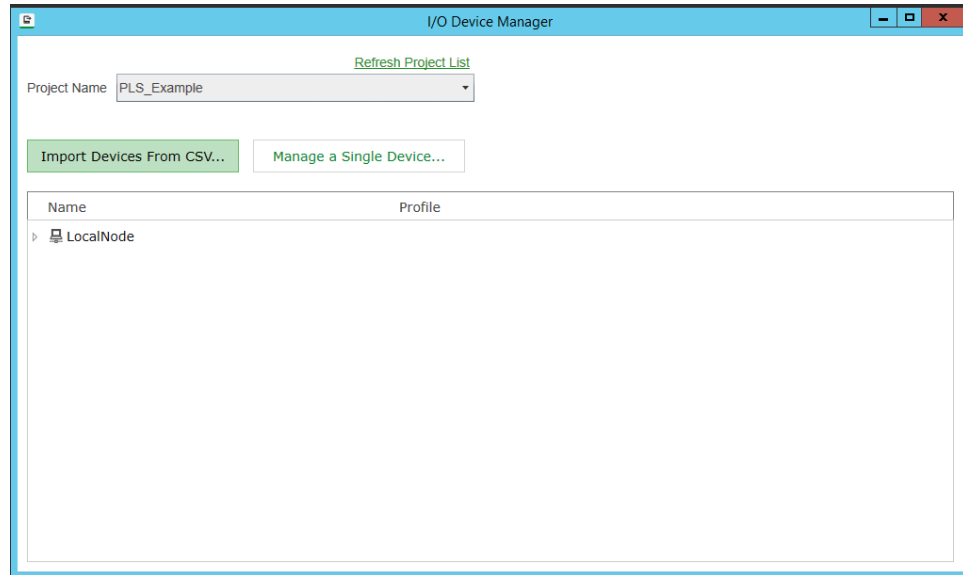
Click Next.

6. 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.
7. 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.
8. 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 I/O Device Manager closes. If you checked the Add/remove ... box, the Welcome screen displays again.
9. From any of the three screens, compile the project.
10. When all errors are corrected, click Run () to view the runtime environment.

## Remove a Single I/O Device From a Project

1. Launch the I/O Device Manager window from the Start menu (Start > Schneider Electric > IO Device Manager).

The I/O Device Manager window displays:



2. Click *Manage a Single Device* , then click Next.  
The wizard displays.
3. Click *Remove one I/O device*, then click Next.
4. At the *Remove profile instance* screen, click the dropdown menu to display the equipment names that were used when the device profiles were previously added to the project. 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 window closes. If you checked the Add/remove... box, the I/O Device Manager window displays again.

## Use a CSV File to Define Multiple Devices

The I/O Device Manager makes it easy to commission a PowerSCADA Expert project. Use this tool to make either single or bulk additions, updates and deletions to the PowerSCADA Expert device database.

Valid communication protocols are:

- DNP3 Serial
- DNP3 Ethernet

- Modbus/RTU Gateway
- Modbus TCP
- ION
- ION/Gateway
- IEC60870-5-104 TCP
- IEC61850

You first need to create the .CSV file that you will use to add the devices. For help, see:

[Create a CSV File to Add Devices on page 129](#)

After you create the .CSV file, you use it to add multiple devices to the project. See:

[Add Multiple Devices to the Project on page 1](#)

### Create a CSV File to Add Devices

Follow these instructions to create a CSV file that you will use to add multiple devices into the project. To view sample CSV files, see:

[CSV File Samples on page 133](#)

**NOTE:** You can edit the CSV file to remove unused columns, or to drag and drop columns to position them where they are easy to read.

If you leave blank a line, the file will only be read to that point. This is useful later, when you want to edit a small number of devices. See [Update One or All Devices in Project on page 132](#)

1. For an existing project: Before you begin, make a backup copy of your project.  
For a new project: In the PowerSCADA Studio, add a new project, define a cluster; and add alarm, trend, and I/O servers (see [Before You Use the I/O Device Manager on page 1](#)).
2. In the Profile Editor, create and export a project that includes the device types and profiles included in this installation. Give the project the same name you used in step 1. (See [Create and Export a Project on page 1](#) for more information.)
3. In Excel, Open Office, or other .CSV file editor, open the example CSV file for your device type. The files are named "exampleXX," where XX is the device type, such as ION or Modbus TCP. These files are located in the Windows Program Data file:  
Program Data > Schneider Electric\PowerSCADA Expert\c8.2\Examples.  
**NOTE:** For sample CSV files, see [CSV File Samples on page 133](#).
4. In the sample CSV worksheet, enter the following information for each device that you want to add:
  - a. ProfileName: the name of the profile that has been exported from the Profile Editor into the target PowerSCADA Expert project. Type the names of the profiles that have been selected for this project. To view names, open the Profile Editor utility.
  - b. Name: Enter the device name, limit of 32 characters; include only letters, numbers, and underscores (\_). The first character cannot be a number or underscore. This field becomes the "Name" on the I/O Devices screen and the "I/O Device" name on the Equipment screen.
  - c. Cluster: The name of the cluster to which the device will be added. If there is only one cluster in the project, this column is not required.
  - d. Equip: Enter the equipment name, limit of 40 characters; include only letters, numbers, and periods (.). The first character cannot be a number or period. This is an optional field. If it's not specified, the "Name" (b) is used. You will use this when adding devices to drawings.

- e. Primary IO Server Name: The name of the primary server I/O server for the device.. If there is only one I/O server in the project, this field is not required.
- f. CommsMethod: Type the communications protocol being used, e.g., MODBUS/RTU via Gateway. See list below for alternate communication connections. When using a composite device, do not use this field. You must enter a "SubProfile1Description" (and a "SubProfile2Description" for the second part of the composite device).

**NOTES:** If the CommsMethod column is missing and you define more than one CommsMethod in the profile:

- If one of them is Modbus/RTU via Gateway, it will be used.

- If one of them is ION it will be used (if there is no MODbus/RTU via Gateway).

If the CommsMethod column is missing and you define only one CommsMethod for the profile, it will be used.

DNP3 Serial  
DNP3 TCP  
Modbus/RTU via Gateway  
Modbus/TCP  
ION  
ION/Ethergate  
IEC60870-5-104 TCP  
IEC61850 Native

- g. PrimaryIPAddress: Type the IP address for the the primary server (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- h. PrimaryEquipmentAddress: Type the device address (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- i. PrimaryPortName: Optional (see description below at *Port Names*.) Type the port name of the primary server (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- j. Standby IO Server Name: If you have a redundant I/O server, type the name here.
- k. StandbyIPAddress: If you have a redundant I/O server, type its IP address.
- l. StandbyEquipmentAddress: If you have a redundant I/O server, type the device address (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- m. StandbyPortName: Optional (see description below at *Port Names*.) If you have a redundant I/O server, type the device port name (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- n. Columns that begin with "SubProfile" followed by a number (e.g., SubProfile1, SubProfile2, SubProfile3, etc.) are used to provide the same information as the Primary and Standby columns for composite devices where each SubProfile is a specific device which is part of the larger composite device.
- o. PrimaryPortNumber: Type the port number of the primary server (required only for MODBUS/RTU and MODBUS/RTU via Gateway).
- p. PrimaryComPort: MODBUS/RTU devices only: 1–99
- q. PrimaryBaudRate: MODBUS/RTU devices only: 1200–115200
- r. PrimaryDataBits: MODBUS/RTU devices only: 1–99
- s. PrimaryStopBits: MODBUS/RTU devices only: 0–99
- t. PrimaryParity: MODBUS/RTU devices only: None, Odd, or Even
- u. Primary ScfFileName: For IEC61850 Native, the file path where the CID (SCL) file is stored.

- v. Primary ledName: For IEC61850 Native, the name of the IED in the CID file. This was created when the profile was added in the Profile Editor.
  - w. FTPHost: For IEC61850 Native, the on-board FTP. Not currently used in PowerSCADA Expert.
  - x. FTPUserName: For IEC61850 Native, the username for FTP on the device.
  - y. FTPPassword: For IEC61850 Native, the password for FTP on the device.
  - z. BRCBS/URCBS: For IEC61850 Native, buffered report control blocks (BRCBs) and unbuffered report control blocks (URCBs) can be used to return data in blocks, rather than in tags. These two fields provide the instruction used for each. The two examples in the example are:  
BRCB: CFG/LLN0\$BR\$BRep01,CFG/LLN0\$BR\$BRep06  
and  
URCB: CFG/LLN0\$RP\$URRep02
  - aa. Optional Parameters: This is optional, and used for functional addressing. This column is a pipe (|) delimited list.
  - ab. Parameter Values: This is optional, and is used in functional addressing. This column includes pipe (|) delimited values for each of the Optional Parameters.
  - ac. Comment: This is an optional description of the device; maximum 254 characters.
5. Close the example .csv file, if it is open.
  6. Go to [Add Multiple Devices to the Project on page 1](#) to add the devices from this .CSV file to your PowerSCADA Expert project.

### Port Names

If you enter a port name, it will be used. Undesignated port names are automatically generated using this format:

**Non/TCP Comms (not having an IP address):**

Px\_BOARDx\_PRJx

where Px = a unique port number, BOARDx = a unique board number, and Prjx = a unique project (e.g., P1\_BOARD3\_PRJ3).

**TCP Comms (having an IP address):**

P(ipaddress)\_BOARDx\_PRJx

where P = port followed by the IP address, BOARDx = a unique board number, and Prjx = a unique project (e.g., P10\_0\_0\_1\_BOARD2\_PRJ3).

To generate the port name:

- For PrimaryPortName, the PrimaryIPAddress is used.
- For StandbyPortName, the StandbyIPAddress is used.
- For SubProfileXPrimaryPortName, the SubProfileXPrimaryIPAddress is used.
- For SubProfileXStandbyPortName, the SubProfileXStandbyIPAddress is used.

### Add a Comment Line

You can add a comment line that will be ignored during processing. To create a comment, begin the line with a double forward slash (//). PowerSCADA Expert skips this line as it processes the device information. See the example below. In the example, lines 5 and 10 will be skipped.

	A	B	C	D	E	F	G	H	I	J
1	ProfileName		Cluster	Equip	PrimaryIC	CommsMe	PrimaryIP	PrimaryEc	PrimaryPc	Primar
2	// These are the meters for Plant 1									
3	PM870 Ful test		c1	equip1	IOServer1	Modbus/F	10.10.10.1	3		P9
4	PM870 Ful test2		c1	equip2	IOServer1	Modbus/F	10.10.10.1	3		P9
5	PM870 Ful	PM870_1	c1	Abcdefgh	IOServer1	Modbus/F	10.10.10.1	3		P9
6	Circuit Mc	CM4000_1	c1	ABCDEFGH	IOServer1	Modbus/F	10.10.10.1	3		P2
7	PM870 Ful	PM870_2	c1	powerme	IOServer1	Modbus/F	10.10.10.1	1		P21
8	Circuit Mc	CM4000_2	c1	monitor2	IOServer1	Modbus/F	10.10.10.1	1		P3
9	PM870 Ful	PM870_3	c1	powerme	IOServer1	Modbus/F	10.10.10.1	2		P21
10	// These are the meters for Plant 2									
11	Circuit Mc	CM4000_3	c1	monitor3	IOServer1	Modbus/F	10.10.10.1	25		P4
12	PM870 Ful	PM870_4	c1	powerme	IOServer1	Modbus/F	10.10.10.1	3		P21
13	Circuit Mc	CM4000_4	c1	monitor4	IOServer1	Modbus/F	10.10.10.1	26		P4
14	PM870 Ful	PM870_5	c1	powerme	IOServer1	Modbus/F	10.10.10.1	5		P21
15	Circuit Mc	CM4000_5	c1	monitor5	IOServer1	Modbus/F	10.10.10.1	27		P4
16										

## Use a CSV File to Define Multiple Devices

The I/O Device Manager makes it easy to commission a PowerSCADA Expert project. Use this tool to make either single or bulk additions, updates and deletions to the PowerSCADA Expert device database.

Valid communication protocols are:

- DNP3 Serial
- DNP3 Ethernet
- Modbus/RTU Gateway
- Modbus TCP
- ION
- ION/Gateway
- IEC60870-5-104 TCP
- IEC61850

You first need to create the .CSV file that you will use to add the devices. For help, see:

[Create a CSV File to Add Devices on page 129](#)

After you create the .CSV file, you use it to add multiple devices to the project. See:

[Add Multiple Devices to the Project on page 1](#)

## Update One or All Devices in Project

If you need to change information for devices in your project (e.g., the IP address changes), do the following. This will update both the CSV file and the project:

1. (optional) If you want to edit only a small number of devices in a very large CSV file, you can save time by moving the devices you are updating to the top of the CSV and leaving a blank line between them and the rest of the CSV file. When PowerSCADA Expert reads the CSV file during the update, it will stop processing after those devices. Make sure you remember to remove the blank line after you update.
2. Make the desired changes to the CSV file. This can include editing existing columns and adding new columns.

**NOTE:** If you change the name of a device, you must add a new column to the CSV file. The name will be either from the Equip field. If you did not specify Equip, the



Name field will be used. For details on CSV fields, see [Create a CSV File to Add Devices on page 129](#).

Name the column OriginalName, and type the device's original equipment name in that column. Otherwise, the update will fail for that device.

3. Open the I/O Device Manager: From the Start menu, click Start > Schneider Electric > IO Device manager.
4. In the I/O Device Manager, click Import Devices From CSV to display the Manage Multiple Devices screen.
5. Click Update Devices.

Changes in the CSV are used to update the devices in the project.

## CSV File Samples

When you want to add multiple devices at a single time, you will create CSV files. The following files are samples of files that may be used for some of the various communication protocols.

For instructions on defining multiple devices, see [Use a CSV File to Define Multiple Devices on page 132](#).

### DNP3 for Serial and Ethernet

	A	B	C	D	E	F	G	H	I	J	K	L
1	ProfileName	Name	Cluster	Equip	Primary IO S	CommsMethod	Primar	PrimaryEc	PrimaryPc	PrimaryPc	Standby I	Stand
2	NewDNP3	DNP_7650Se	c1	DNP3_meterSerial	IOServer1	DNP3 Serial		101		Port4		
3	NewDNP3	DNP_7650	c1	DNP3_meter	IOServer1	DNP3 TCP	10.167	100	20000	Port5		
4												
5												
6												
7												
8												
9												
10												
11												

### IEC104.2

	A	B	C	D	E	F	G	H	I	J	K
1	ProfileName	Name	Cluster	Equip	Primary IO Serve	CommsMethod	Primary	PrimaryEc	PrimaryPc	PrimaryPc	Stand
2	IEC104Device	Device104	c1	IEC104New	IOServer1	IEC60870-5-104 TCP	10.171.1	3		port7	
3											
4											
5											
6											
7											
8											
9											
10											
11											
12											
13											
14											

### IEC61850

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	ProfileName	Name	Cluster	Equip	Primary IO Server Name	CommsMethod	PrimaryScfFileName	PrimaryIedName	FTPHost	FTPUserName	FTPPassword	BRCBS	URCBS
2	SEL3515	DEV850	c1	SELDevice	IOServer1	IEC61850 Native	C:\ProgramData\Sch SEL_3515_1		10.10.1.1	XXXXXXXX	YYYYYYY	CFG/LL	CFG/LLN
3													

## Use PowerSCADA Expert One-Lines

Click the following links for related one-line information.

[One-Line Flowchart on page 134](#)

[What Are PowerSCADA Expert One-Lines? on page 135](#)

[One-Line Device \(zOL\) on page 136](#)

[One-Line Colors on page 137](#)

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 137](#)

[Start and Stop One-Lines on page 139](#)

### 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 144](#)

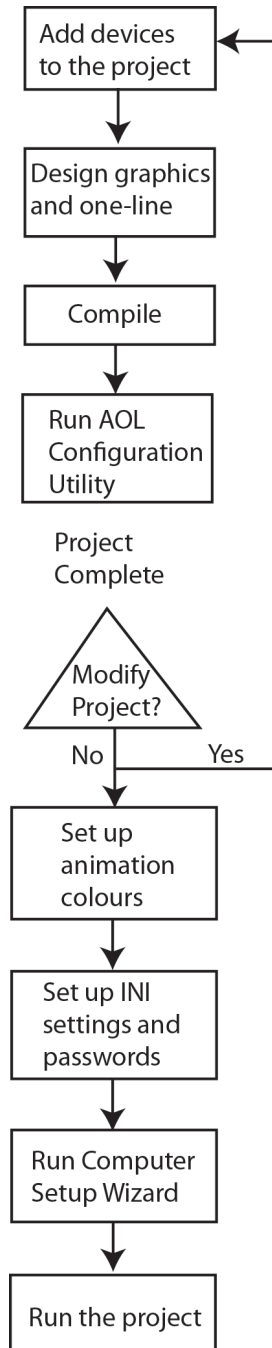
[Perform Repairs for One-Lines on page 147](#)

Setting Up Animation Colors:

[One-Line Colors on page 137](#)

INI Settings and Passwords:

[Add INI Settings to AdvOneLine.ini and Citect.ini on page 137](#)



After you run the project, ensure that the password is encrypted (see *IsEncrypted* in [Add INI Settings to AdvOneLine.ini and Citect.ini](#) on page 137).

Ensure that all steps in the Quick Start procedure have been followed: .

## What Are PowerSCADA Expert One-Lines?

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

- graphically propagate unknown conditions downstream through a one-line diagram. When there is a communications loss to a device, the one-line provides accurate system status, even though information is not available.

- eliminate "seesaw logic": when bus status temporarily toggles on and off after an upstream breaker opens
- handle the continuation of a one-line diagram when it extends from one page to another
- provide built-in support for power flow diagrams; one-line color "animates," based on the source that is feeding the circuit.

### Where are the files located?

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

<b>PLS_Include</b>	
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
<b>Current RUN project</b>	
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 I/O Device Manager.
2. Click *Create an I/O Device in the project*. Click *Next*.
3. Select the device called *OneLine Device Setup*. Click *Next*.
4. Follow the remaining steps to add the device.

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

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

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


## One-Line Colors

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

In PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, coloring is based on the source and meter line active states. Sources dictate the colors for each genie. Meters can only determine if a bus is active. When the bus is live, the meter then colors based on the source that is connected to the bus. If there is no source, the default color is used.

**NOTE:** Depending on how you configure transformers, you can either use this "pass-through" coloring, or you can use "voltage-level" coloring. See [Configure a Transformer on page 142](#) for more information.

### Assigning Colors

To assign a color to a source, open the Color Configuration tab of the One Line Configuration Utility (PowerSCADA Studio, Projects tab, click *'Launch the Single Line Configuration Utility'* ().

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

## Add INI Settings to AdvOneLine.ini and Citect.ini

NOTE: See [Application Configuration Utility on page 220](#) for screens that streamline configuration.

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

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

### AdvOneLine.ini Settings

The following parameters are set in the Application Configuration Utility ([One-Line Engine on page 224](#)): UpdateInterval, PrimaryServerIP, StandbyServerIP, HealthTimeout, MaxStartupDelay, LoginUserName, LoginPassword, and LogFileLength.

Parameter	Description	Default Value
Updateinterval	The interval at which the system tries to solve the system one-line. This interval can be changed to slow down the rate at which the animation is solved. Specifying a rate faster than possible will force the engine to solve the system as quickly as possible.	1000 msec

Parameter	Description	Default Value
PrimaryServerIP	Used by redundant configurations to specify the IP address of the server on which the primary IO Server resides. This parameter is required for a redundant configuration. If either the primary or standby IP addresses are not specified, the logic engine will assume that the system is not redundant.	N/A
StandbyServerIP	Used by redundant configurations to specify the IP address of the server on which the standby IO Server resides. This parameter is required for a redundant configuration. If either the primary or standby IP addresses are not specified, the logic engine will assume that the system is not redundant.	N/A
HealthTimeout	This is a performance parameter that dictates that amount of time that must elapse before the one-line engine is considered non-functioning, and a PC-based alarm is raised in PowerSCADA.	[Updateinterval] + [TagSubscribeWait] * 5 milliseconds Minimum value: 1000 msec
DefaultColor	This cosmetic parameter tells the engine the default color to be assigned to objects on the screen at system startup. This is useful for identifying components that have been left out of the CSV or simply as a means of having the engine set all currently unused objects to a color that indicates that they are not being monitored. If an invalid color is specified, the engine will default to black.	250
MaxStartupDelay	Sets the amount of time the AdvOneLine.exe has to start up. If this time is exceeded, initial tag subscriptions will not succeed, and the EXE will report an exception.	60 sec
StartupDelay	Sets the amount of time after AdvOneLine.exe has started for the system to be on line and all initializations complete.	[Updateinterval] + [TagSubscribeWait] * 5 milliseconds Minimum value: 1000 msec
LoginUserName	This is the user name to be used for the ctAPI connection in AdvOneLine.	aol
LoginPassword	This is the user password to be used for the ctAPI connection in AdvOneLine.	aol
IsEncrypted	Determines if the password is encrypted. The first time the project is opened in run time, the password is automatically encrypted, and this will be set to True.	False (changed to True after the first run and successful password encryption)
CitectIniPath32	Provides the path to the global Citect.ini file for a 32-bit operating system install. This setting must be changed if SCADA is not installed on the C: drive, or if the Citect.ini file is moved/installed in another directory.	Default value: C:\Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Config
LogFileLength	Suggests the log file length in number of lines. After surpassing this limit, the log file is saved with suffix ".bak," and a new file is created.	Default value: 5000 Allowed values: 10–10000
DebugLevel	Sets the level of logging for AdvOneLine.exe. Multiple values are separated by   (e.g., Error Warn).	N/A Allowed values: All, Error, Warn, Debug

### Citect.ini Settings

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

### Start and Stop One-Lines

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

Only call these functions on an I/O server that is communicating with an [One-Line Device \(zOL\) on page 136](#).

**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 139](#)

[Configure an Automatic Transfer Switch \(ATS\) on page 140](#)

[Configure a Busbar on page 140](#)

[Configure a Circuit Breaker or Switch on page 141](#)

[Configure a Meter on page 141](#)

[Configure a Source on page 142](#)

[Configure a Transformer on page 142](#)

[Enable Lockout/Tagout on page 113](#)

[Create New Genies on page 115](#)

[SupportedGenies.xml File on page 143](#)

#### Genie Type Descriptions

In the One Line Configuration Utility, there are several genies. Each one has a certain defined list of information that displays by default. You can also expand this list to

include everything that is known about that genie type. To learn about each genie's data, click one of the following links:

[Breaker and Switch Information on page 150](#)

[Meter Information on page 151](#)

[Source Information on page 151](#)

[Transformer Information on page 152](#)

For configuration instructions, see:

[Configure a Busbar on page 140](#)

[Configure a Circuit Breaker or Switch on page 141](#)

[Configure a Meter on page 141](#)

[Configure a Source on page 142](#)

[Configure a Source on page 142](#)

[Configure an Automatic Transfer Switch \(ATS\) on page 140](#)

See also:

[Enable Lockout/Tagout on page 113](#)

[SupportedGenies.xml File on page 143](#)

### Configure an Automatic Transfer Switch (ATS)

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

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

#### Transfer Switch Information: Left/Right/Bottom

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

#### Display Information

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

For information on how colors are determined, see [One-Line Colors on page 137](#).

### Configure a Busbar

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

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



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

For information on how colors are determined, see [One-Line Colors on page 137](#)

### 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 I/O Device Manager, e.g., *CM4Main*). To view the equipment that is available for this page, you must have the project selected in the PowerSCADA Studio.
3. **Breaker/Switch Number:** Type the number you want to use for this breaker or switch. Valid entries: 1–1000. This number must be unique within this one-line.
4. **Source** and **Destination:** When you connect to busbars, type the numbers for the connection source and destination busbars. Valid entries: 1–1000.
5. **Label 1** and **Label 2:** Type the information that you want to appear in the upper left corner of the genie in the runtime environment.
6. **Value Type:** From the drop-down list, choose the tag you want to use for this genie. The related information displays in the lower left corner of the genie in the runtime environment.

This tag causes real-time data (such as *currents*) to display on the genie status page in the runtime environment. If you do not choose a tag here, the status page will not display the real-time data. See [Use the Equipment Pop-Up Page on page 180](#) for more information.

7. **Units:** From the drop-down list, choose the unit that you would like to display on the genie in the runtime environment. Units that display here were added in the Profile Editor (see [Add or Edit a Base Engineering Unit or Conversion on page 40](#)).
8. **Multiplier:** Enter the multiplier that is to be applied to the units chosen in step 7.
9. Click OK to save the genie to the page and to return to the graphics builder page.

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

For information on how colors are determined, see [One-Line Colors on page 137](#).

### 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 I/O Device Manager, e.g., *CM4Main*). To view the equipment that is available for this page, you must have the project selected in the PowerSCADA Studio.

3. **Meter Number:** Type the number you want to use for this meter (valid entries: 1–1000). This number must be unique within this one-line.
4. **Busbar Number:** Type the number for the busbar that connects to the meter. Valid entries: 1–1000.
5. **Line Active:** Enter the appropriate Cicode expression (such as *MyTag1 > 0*) to determine when the meter detects power on the busbar. See "How do I create and configure busbars" in [Appendix 5: Frequently Asked Questions \(FAQs\) on page 293](#) for more information.
6. **Label 1** and **Label 2:** Type the information that you want to appear in the upper left corner of the genie in the runtime environment.
7. **Value Type:** From the drop-down list, choose the tag you want to use for this genie. The related information displays in the lower left corner of the genie in the runtime environment.

This tag causes real-time data (such as *currents*) to display on the genie status page in the runtime environment. If you do not choose a tag here, the status page will not display the real-time data. See [Use the Equipment Pop-Up Page on page 180](#) for more information.

8. **Units:** From the drop-down list, choose the unit that you would like to display on the genie in the runtime environment. Units that display here were added in the Profile Editor (see [Add or Edit a Base Engineering Unit or Conversion on page 40](#)).
9. **Multiplier:** Enter the multiplier that is to be applied to the units chosen in step 8.
10. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 137](#).

### Configure a Source

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

1. Display the source window by pasting the generator or utility genie on to the graphics builder page.
2. **Source Number:** Type the number you want to use for this source (valid entries: 1–100). This number must be unique within this one-line.  
The source number is used when you determine coloring for the one-line. For more information on this, see [One-Line Colors on page 137](#).
3. **Busbar Number:** Type the number for the busbar that connects to the source. Valid entries: 1–1000.
4. **Line Active:** Enter the appropriate Cicode expression (such as *MyTag1 > 0*) to dictate when the source powers the busbar. See "How do I create and configure busbars" in [Appendix 5: Frequently Asked Questions \(FAQs\) on page 293](#) for more information.
5. Click OK to save the genie to the page and to return to the graphics builder page.

For information on how colors are determined, see [One-Line Colors on page 137](#).

### Configure a Transformer

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

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

2. **Top and Bottom Source Number:** For each source, whether top or bottom, type a source number to control voltage-level coloring for the secondary side busbar. Valid entries: 1–100.

There are two possible configurations:

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

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

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

For information on how colors are determined, see [One-Line Colors on page 137](#).

### GenieDefinitions.xml File

#### [Example](#)

Use the GenieDefinitions.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 143](#)

### SupportedGenies.xml File

#### [Example](#)

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

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

See also: [GenieDefinitions.xml File on page 143](#)

## 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 144](#)

[Review Genie Configurations on page 144](#)

[One-Line Errors and Warnings on page 145](#)

[Perform Repairs for One-Lines on page 147](#)

[Automatic Transfer Switch \(ATS\) Information on page 148](#)

[Busbar Information on page 149](#)

[Breaker and Switch Information on page 150](#)

[Meter Information on page 151](#)

[Source Information on page 151](#)

[Transformer Information on page 152](#)

## One-Line Configuration Utility

To access this screen from the PowerSCADA Studio. From the left hand panel, click 'Launch the Single Line Configuration Utility' (  ). There are two tabs.

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

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

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



On the **Color Configuration tab**, you can assign colors to sources. see [One-Line Colors on page 137](#) 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 left panel of the PowerSCADA Studio, click 'Launch the Single Line Configuration Utility' (  ).

Click OK to the message that displays.

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 PowerSCADA Studio.
Show By:	Type: Information is sorted first by genie type, then by page. This option is useful when you want to see all genies of a certain type together, regardless of where they are in the drawing pages.  Page: Information is sorted first by page, then by genie type. This option is useful when you want to see all genies on a certain page.
Advanced Properties	Check this box to view the basic information plus any additional information relevant to that genie type.
State Filters:	Check the individual boxes for how you want to view information. For example, you might only be interested in viewing genies that have error states. This option controls only the genie information in the right-hand pane.
Genie Types	Types are: breakers, busbars, meters, sources, and transformers
Genie Information	Columns of information display: In the Basic (default) view: the most used information If you click Advanced Properties, you see the basic information, followed by all of the information known about the genie(s) you are viewing.
Repair—Upgrade Project	Check this box to cause the repair feature to repair the entire project.  Use this feature only to upgrade projects that are earlier than PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. This option repairs the entire project, renumbering all busbars, breakers, meters, duplicate Sim sources, and sources. Additionally, busbar line active states are used to determine meter and source line active states.  DO NOT perform Repair—Upgrade Project more than once, and do not perform it on a PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards 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 150](#)

[Busbar Information on page 149](#)

[Meter Information on page 151](#)

[Source Information on page 151](#)

[Transformer Information on page 152](#)

### One-Line Errors and Warnings

Typical one-line errors are:

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

## Communication Errors

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

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

## Error Logging

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

Error Message	Description
Main Execution Loop Unexpected Failure	The main logic loop has thrown an exception that has not been handled by other error messages.
AdvOneLineDebugBus.Csv is locked	Another process or user has this required CSV file locked. Ensure that you do not have the file open.
PowerSCADA ExpertRunning Project Path: "PATH" Does not Exist. Please Shutdown your Project and Try Running your Computer Setup Wizard	The Citect.ini "Run" parameter has an invalid project path that does not exist. Run the Computer Setup Wizard, and this path should be corrected.
PowerSCADA ExpertRunning Project Pth Not Specified. Please Shutdown your Project and Try Running your Computer Setup Wizard	This problem is almost exclusively caused by not running the Computer Setup Wizard.
PLSCADA is not in runtime	You must have your project running before you execute the AdvOneLine.exe file.
Failed to Establish Connection with CTAPI. PLSEngine.establishPLSCConnection (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.

### CAUTION

#### DATA CORRUPTION

Always make a backup copy of your project before you perform a repair.

**Failure to follow these instructions can result in data loss and corruption of your project.**

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

1. Open the configuration utility: From the Start menu, click All Programs > Schneider Electric > Configurator.

The first time you launch this utility, it could take several minutes for it to read all of the project information.

The utility will read the project's graphics pages to analyse the project. If it appears that the project is from a version prior to PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, a message displays, telling you that the project appears to be out of date. The default "Upgrade Project" is checked in the lower right corner.

2. Click OK.

Genie information for the selected project displays. For descriptions of the fields on this page, see *Review Genie Configurations*.

3. Choose the type of repair you want to perform:

**Repair option alone** (Upgrade Project not checked) attempts to fix errors and warnings in a project (used for PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards).

**Repair option with Upgrade Project checked** is used to upgrade projects from previous versions of the product. This option renumbers all genies in the project. Do not perform this option on a project more than once, and do not perform it on PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards projects.

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

The following table describes the repairs made in each option.

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

4. Click Repair.

You see a message that describes the degree of repair that is about to take place.

Each message says that graphics pages "will not be modified by the repair process." This means that the repairs will not be applied to your project graphics pages until you press Save.

5. Click Yes to initiate the repair option that you have selected.

A Repair Summary window displays, listing the repairs that have been initiated.

6. To save a .csv copy of this summary, click Export. At the Save As window, type a file name and choose the location at which you want to save the file.

7. Click OK.

The genie information changes, indicating that the repairs have been made.

8. Click Save.

The Save window appears. This is where the changes are saved to your project.

9. Click Yes to save the changes to the graphics pages of the project.

Or click No to back out of making the changes.

If you click No, click Close, then click No when you are asked whether you want to save the modified project.

If you click Yes, the changes are saved to the project. For a large project, this might take several minutes. When the repairs are saved to the project, you see a Save Summary window, which lists the repairs that were made and saved.

Click Export to save a .csv file of these changes. Click OK to exit the summary window and return to the One Line Configuration Utility window.

10. It is possible that some errors and warnings will not be repaired, for example, missing busbar numbers or missing equipment. Click individual errors or warnings to view them (note that the warning and error icons include a tooltip to tell you what is wrong). Note the missing information, then go to the graphics builder to make the necessary changes.

11. Compile the project and then run it.

### Automatic Transfer Switch (ATS) Information

#### ATS Information

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

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

Basic ATS information includes:

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



Column	Description
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
<b>Errors (✖)</b>	
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
<b>Errors (✖)</b>	
Busbar number must be a number greater than 0.	The busbar number is missing, or it is less than or equal to 0. Add or change the busbar number.

### Breaker and Switch Information

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

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

Basic breaker information includes:

Column	Description
State	Normal (✔), Warnings (⚠), or Errors (✖). See the following table for explanations of errors.
Page	Name of the page on which the genie is found (displays only from the folder level).
ID	This is the breaker number, assigned when adding it to a page of a one-line.
Equipment	The equipment name entered when adding the genie via the I/O Device Manager.
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
<b>Errors (✖)</b>	
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 I/O Device Manager.
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
<b>Errors (✖)</b>	
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.
<b>Warnings (⚠)</b>	
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
<b>Errors (✖)</b>	
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.
<b>Warnings (⚠)</b>	
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.

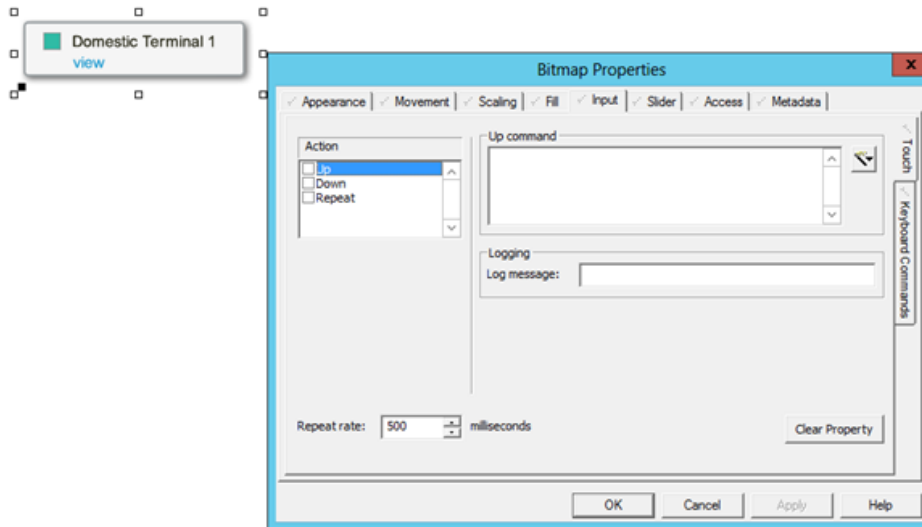
Errors that you might see for transformers are:

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

## Add Custom Images to Graphics Pages

A user might want to add custom images developed in external imaging software to a graphics page in PowerSCADA Expert. One example might be a floor plan of a facility or an image depicting a physical piece of switchgear, such as an elevation drawing.

1. Create you image in your imaging editor of choice. Several file types are supported, but bitmaps tend to look the best when added to PowerSCADA Expert.
2. In the PowerSCADA ExpertGraphics Builder, select File >Import and select your image.
3. Re-size the image as necessary.
4. Overlay the image with any genies or data display objects that need to be displayed.
5. (Optional). If the image is intended to function as a button, it can be configured to do so by right-clicking and configuring properties similar to configuring any buttons or images that are native to PowerSCADA Expert. As an example see the imported bitmap below and the properties that can be configured such that the bitmap acts as a button in the runtime environment.



For more information see the Citect PC-Based help topics:

- Using PowerSCADA Expert > Graphic Pages > Configure Graphic Objects > Import Dialog Box
- Using PowerSCADA Expert > Graphic Pages > Configure Graphic Pages > The Bitmap Editor

## Work with Alarms

In this section, you will find these topics:

[Alarms Overview on page 154](#)

[Set Up Alarm Pages on page 155](#)

[Add Setpoints and Delays on page 155](#)

[Set Up an Alarm Based on an Enumeration on page 155](#)

[Change an Alarm Severity on page 156](#)

[Enable Waveforms for Onboard Alarms on page 156](#)

[Set Parameters for Event Log Length and Historical Logging of Events on page 156](#)

[Add an Onboard Alarm Tag on page 157](#)

[Set Up Audible Alarms on page 157](#)

## Alarms Overview

This section discusses two alarm types: time stamped analog and time stamped digital. To access the alarms: From PowerSCADA Studio, select the System Model tab, then click Alarms. Select the alarms to edit; in the right-hand pane, then click Alarms. In the right-hand pane, the alarm types display. Double-click the one you want to view/edit.

### PC-Based Alarms

1. PC-based alarm tags are added in the Profile Editor, when adding each device profile. See [Add Edit or Delete Device Profile on page 59](#) for instructions. For instructions on entering setpoints and delays, see [Add Setpoints and Delays on page 155](#).

## 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 157](#).

## 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 the PowerSCADA Studio, click Visualization tab > Menu Configuration.
2. In the Menu Command column, add the Cicode method that will open the page:

```
PLSDspShowAlarm(INT nType)
```

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

Example (for disabled alarms):

```
PLSDspShowAlarm(3)
```

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

## Add Setpoints and Delays

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

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

- From the Analog Alarms window (accessible from the PowerSCADA Studio), you can type the setpoint and delay values for each alarm.
- When in the runtime environment, you can edit setpoints/delays that were set by the method above. See [Use the Equipment Pop-Up Page on page 180](#) 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 84](#).

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 180](#) for instructions.

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

You can use two parameters to determine the maximum number of entries in the Event Log and whether you want to log entries after they are FIFO'd out of the Event Log.

### Event Storage: [Alarm]SummaryLength Parameter

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

The default value is 5000.

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



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

## Add an Onboard Alarm Tag

When a device onboard alarm has not been included in PowerSCADA Expert, you can add it from the Profile Editor. You need to follow these steps to include the device's unique identifier. Otherwise, the alarm will not annunciate in the Graphics page.

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

1. From the device, obtain the unique identifier for this alarm. Additionally, for MicroLogic, you need to include the unique sub-identifier.
2. You also need the file number in which alarms are stored on the device.
3. From the Profile Editor, add the onboard alarm (see [Onboard Alarm Tag Addresses on page 53](#) 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 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 I/O Device Manager, 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

## Set Up Tag Viewer

The tag viewer is a feature that displays in the graphics page during runtime . Use the tag viewer to view details about equipment. This screen provides the status of project tags.

To add the tag viewer to a project graphics page, follow these steps:

1. From the PowerSCADA Studio, click Visualization tab > Menu Configuration.
2. In the Menu Command line, add the Cicode method that will open the page:

```
PLSPageDisplay ("PLSTagView")
```

When viewing the tag viewer in runtime, as long as the screen resolution is one that PowerSCADA Expert supports, the view will be correct.

For information about viewing tags, see [View the Tag Viewer on page 184](#).

## Restore and Back Up Projects

In this section, you will find these topics:

[Restore a Project on page 158](#)

[PowerSCADA Expert Backup on page 158](#)

### Restore a Project

To restore a project, overwriting its current settings, follow these steps:

1. From the PowerSCADA Studio, Projects tab, click Backup > Restore;, select the project you want to restore; click Toos > 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 aPowerSCADA Expert project file. To back up a Profile Editor project file, see [Profile Editor Export on page 74](#).

1. From the PowerSCADA Studio, Projects tab, click 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 sub-directories." This saves the ProfileInstanceConfigurations file, which is used in upgrades.
5. In the Options box, check "Save configuration files." This saves the citect.ini file.
6. Click OK.

The backup .CTZ file is written to the location that you choose during backup. This is a 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 248](#).

For information about other parameters, see the Cicode Reference help file in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## Use Cicode in PLS\_Include

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

Click to view descriptions/instructions for each module:

[PLSProviderEngine.ci Module on page 159](#)

[Clear Cache and Refresh Platform on page 161](#)

### PLSProviderEngine.ci Module

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

#### Module Construction

The following string functions are included in this module:

#### CallProvider

This function invokes a provider (whose GUID-based identifier must appear in the sProvider argument) with a single string as input (the sArgs argument). The input string can consist of anything that is meaningful to the provider that you invoke.

The provider then returns a string-based token.

Construction of CallProvider:

```

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

```

## GetProviderStatus

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

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

Construction of GetProvider Access:

```

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

```

## GetProviderResult

This function retrieves the result from a provider. Pass a unique token (obtained previously by calling CallProvider) to this function. It returns the provider result as a

string. Note that you should only call this function after you verify that the provider work is 100 percent complete.

Construction of GetProviderResult:

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

## Clear Cache and Refresh Platform

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

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

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

Each of these methods uses the PLSProviders.ci module. See [PLSProviderEngine.ci Module on page 159](#) for instructions on creating the statements needed.

### PLS\_ClearCache

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

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

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

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

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

```
PLS_ClearCache();
```

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

- If the Schneider Electric CoreServiceHost is on a different machine, you must supply parameters to identify where the Application Services core resides. For example, if the customer's Schneider Electric CoreServiceHost resides on an IOserver named "IOserver1" on "Cluster1", to call PLS\_ClearCache, enter:

```
PLS_ClearCache("IOserver", "IOserver1", "Cluster1");
```

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

### PLS\_PlatformRefresh

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

To run the refresh, call the PLS\_PlatformRefresh function by doing one of the following:

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

```
PLS_PlatformRefresh();
```

- If the Schneider Electric CoreServiceHost is on a different machine, you must supply parameters to identify where the Application Services core resides. For example, if the customer's Schneider Electric CoreServiceHost resides on an IOserver named "IOserver1" on "Cluster1", to call PLS\_PlatformRefresh, enter:

```
PLS_PlatformRefresh("IOserver", "IOserver1", "Cluster1");
```

## Customize Default Behaviours

In this section, you will find these topics:

[Time Zone Settings on page 162](#)

[Time Synchronization on page 163](#)

[Trend Tag Scan Intervals on page 163](#)

[Disk Storage Calculation for Trends on page 164](#)

[Deadbands and Ignored Devices and Topics on page 164](#)

To learn about the parameters that you can use to configure drivers, see [Appendix 2: Citect INI Parameters on page 248](#).

## Time Zone Settings

The system allows you to interact with devices located in a different time zone than the PowerSCADA Expert I/O Server. To do this, the system converts any alarm/waveform timestamp as well as the actual time sent within the Set Time command from / to the local time zone. In order to take into account daylight savings properly, the Windows time zones database is used. Thus, time zone names must be taken directly from this database (case-insensitive), otherwise the system will default to using the I/O server's local time zone. The Windows time zone database is located in the Windows registry in HKEY\_LOCAL\_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\Time Zones\. Examples of time zone names are

- AUS Central Standard Time
- China Standard Time
- Pacific Standard Time (Mexico)

Time zones for remote devices can be specified on two levels:

1. Use section [ProtocolName.ClusterName.PortName.IODeviceName] to specify the time zone for a particular device, e.g.  
[PLOGIC870.Cluster1.PM870\_Port.PM870\_Device1]  
Timezone = Singapore Standard Time
2. Use general section [POWERLOGICCORE] to specify the time zone for all devices, e.g.  
[POWERLOGICCORE]  
Timezone = Mountain Standard Time

The device-specific time zone specification takes precedence. In other words, if both of the above examples are present in the Citect.ini file, the PM870\_Device1 would be located in "Singapore Standard Time" time zone, and all the other I/O devices in the project would be located in "Mountain Standard Time" time zone.

If there is no time zone specification (or if it does not match the time zone from Windows database), the device would be in the same time zone as the machine where the I/O Server is running; thus, no time conversion will be done.

If only the first of the above examples is present within the Citect.ini file, the PM870\_Device1 would be located in "Singapore Standard Time," and all the other devices use the current local time zone.

## Time Synchronization

Current time can be sent to the corresponding device by means of Set Time command or (in case of Sepam) by writing directly to the corresponding registers within the device. In addition to the manual procedure, this process can be scheduled to occur periodically (using PowerSCADA Expert events).

Non-manual time synchronization causes the Set Time command to be sent automatically, based on a device state or event originating from within the device.

Automatic time synchronization applies only to Micrologic and PM devices and takes place based on the following rules:

1. For Micrologic devices, the value of the top-most bit of the register 679 is examined (for both the Circuit Breaker Manager and the Chassis Manager). If the bit is equal to 1, it means that the device is out of sync and needs to be synchronized.
2. For PM devices, an alarm 50700 ("Unary Power Up / Reset") indicates that the device needs to be synchronized. In addition, bit 6 of register 3055 of the device is examined. If this bit is equal to 1, the device has a real-time clock; so automatic time synchronization should never take place.

## Trend Tag Scan Intervals

When you select a trend tag for a device profile (Add/Edit Device Profile screen), the tag will be scanned at the "long" interval" (every 15 minutes, with FIFO storage of 12 months); but certain trend tags have an additional "short" scan interval. This interval is set by default at 5 seconds, with FIFO storage of two weeks.

The default tags are: Current A, Current B, Current C, Voltage A-B, Voltage B-C, Voltage C-A, Power Factor Total, Apparent Power Total, Reactive Power Total, Real Power Total, and Frequency. When you choose one of these tags for trending, you will get both long and short interval trending. The long interval trend will use the trend tag name from the Profile Editor. The short interval trend tag will have the same name as the long tag with an "s" appended to it.

You can edit the *Profile Editor.exe.config* file to add or delete tags that will have short scan intervals, and to change the short scan interval for all of the tags that are listed.

To edit short scan interval settings, follow these steps:

1. Locate and open *Profile Editor.exe.config* in Notepad:  
[Project Drive]\Program Data\ Schneider  
Electric\PowerSCADA Expert\v8.2\Applications\Profile Editor
2. To change the short scan interval, scroll the "TrendShortIntervalSamplePeriod" setting. The default value is 00:00:05, or five seconds (HH:MM:SS). Changing this rate will change the interval for all of the tags that are listed in the setting in step 3.
3. To change the tags that are included in the short scan interval, scroll to the "TrendShortIntervalTags" setting. The numbers listed (defaults: 1003,1004,1005,1050,1046,1042,1014,1015,1016,1001,1034) are the tag IDs. You can add or delete tags. Tag IDs are listed on the Define Device Type Tags tab (when the Advanced Properties option checked).

**NOTE:** If you choose a device that includes the tags in this list, you will always have these short scan interval tags included.

For example, if you wanted to change the scan interval to ten seconds and add Overcurrent A for a CM4000, you would edit these two lines in this way:

```
"TrendShortIntervalSamplePeriod" value="00:00:10"
```

```
"TrendShortIntervalTags"  
value="1003,1004,1005,1050,1046,1042,1014,1015,1016,1001,1034,19"
```

## Disk Storage Calculation for Trends

There are two methods of calculating disk space usage for trends: scaled and floating point. The Profile Editor uses floating point by default. For more information on these calculations, see **Calculating Disk Storage** in the *citectSCADA.chm* help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## Deadbands and Ignored Devices and Topics

These two features allow you to limit information that you see in system queries and data acquisition. You set the limits for these features in the *Configuration.xml* file (C:\Program Files (x86) > Schneider Electric > PowerSCADA Expert > v8.2 > Applications > AppServices > bin > Configuration.xml).

### Deadbands

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

```
    <Value />
```

```
</ConfigurationItem>
```

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

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

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



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

### Ignored Devices/Ignored Topics

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

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

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

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

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

Save your changes.

## Waveform Management

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

[Waveform Storage on page 166](#)

[Waveform Database and Special Waveform Tags on page 166](#)

### 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_0000000001203566197_0000000511_utc.CFG
DST_0000000001203566197_0000000511_utc.DAT
```

Waveform DB root path is configured in the WaveformDB configuration section.

NOTE: in case of redundant I/O devices, only the name of the primary I/O device will be used when waveform storage path is constructed.

The CFG file is a **Comtrade configuration file**, and the DAT file is the **Comtrade data file**. Within the CFG file is a timestamp that reflects the device time start time of the waveform (this time is not adjusted to the I/O Server time zone or daylight saving, but it is stored per the device configuration). The file name has the UTC time in seconds since 1970 of the waveform.

The prefix of waveform file name reflects the type of the waveform. Currently, waveforms of the following types are supported:

DST_	Disturbance waveform
ADT_	Adaptive waveform
SST_	Steady state waveform

If it is detected that the waveform data file has changed while it is being downloaded, the file gets discarded and is not stored on the IOserver.

### Waveform Database and Special Waveform Tags

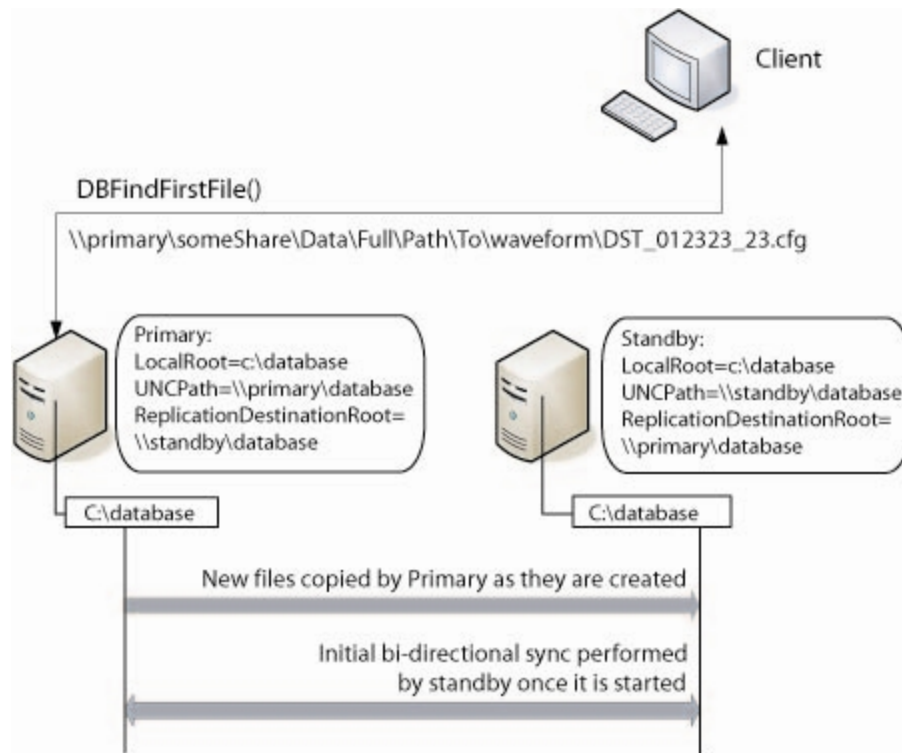
PowerSCADA Expert allows you to browse the waveform database for specific I/O devices. Search for all waveforms within certain time frame is also supported, allowing you to search for all waveforms that could be linked with a given alarm. When you

perform this search, a list of all matching waveforms displays. If there are multiple waveforms in the list, you can select the waveform you want to view.

In addition, there are two special digital waveform tags defined (0 = FALSE, 1 = TRUE):

- **WaveformDownloading**: indicates whether a waveform file is currently being downloaded
- **WaveformCollectionEnabled**: indicates whether the waveform collection is enabled at all

The figure below illustrates a configuration example and replication and linkage processes.



# The Runtime Environment

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

*Open Firewall Ports for PowerSCADA Expert Runtime* on page 169

*Launch and Close the Runtime Environment* on page 169

*View the Graphics Page in the Runtime Environment* on page 170

*Log On to the Runtime Environment* on page 170

*View the Interface* on page 170

*View the Alarms/Events Page* on page 172

*When Alarms do not Display Correctly* on page 174

*Event/Alarm Log Columns Table* on page 175

*Alarm/Event Filter Form* on page 176

*Use the Analysis Page* on page 179

*Use the Equipment Pop-Up Page* on page 180

*Perform IEC 61850 Advanced Control* on page 185

*EcoStruxure WebServices (EWS)* on page 186

*Use Basic Reports* on page 187

*Set up the Runtime Environment for Basic Reports* on page 187

*Set Up a Display Client for Basic Report Viewing* on page 188

*Tasks in the Basic Reporting Application* on page 188

*Single Device Usage Reports* on page 196

*Multi Device Usage Reports* on page 197

*Tabular Reports* on page 197

*Trend Reports* on page 198

*Enable Windows Authentication for Basic Reporting* on page 198

*URL Routing for Basic Reports* on page 199

*Localization for Basic Reports* on page 200

*Use LiveView* on page 200

*LiveView Viewer* on page 207

*Set Up LiveView* on page 209

*LiveView Formulas* on page 213

*LiveView Placeholders* on page 212

*LiveView Thresholds* on page 214

*LiveView Formatting* on page 211

*Multi-Monitor Support* on page 219

*The Demo Mode* on page 219

*Application Configuration Utility* on page 220

*Application Services* on page 221

*Application Services Host—Citect Data Platform* on page 223

*Applications* on page 223

*Diagnostics Page/I/O Device Settings* on page 225

*Single Sign On* on page 235

## Open Firewall Ports for PowerSCADA Expert Runtime

For the system to properly run, you need to ensure that the following ports are properly set.

Before you begin, define the primary and standby alarm servers, trend servers, and I/O servers. Then, to enable communication for runtime operations, use the information in the following tables. Each server has a unique default port assigned to it. Use this default port only with that type of server. If you attempt to use a default port on another type of server, you will see a compilation error:

Invalid port number (2073-2082,20222,21) are reserved.

### Default Port Numbers and Associated Server Types

Default Port	Legacy Port	Server Type	Server Role
21	N/A	FTP Server IDC	Page downloads for IDC Internet Display Server/Client communications
2073	N/A	CTAPI	CTAPI communications
2074	N/A	Client	Cicode debugging
2084	2075	Reports server	Reports server communications
2080	2076	Alarm server	Alarm server communications
2085	2077	Trends server	Trends server communications
2078	N/A	I/O server	Legacy I/O communications (version 6 or earlier)
2080	2076	Alarm server	Alarm properties connector
2082	2078 (client server comms only)	I/O server	Publish, subscribe I/O server communications
20222	N/A	ODBC	ODBC server
5482	N/A	Alarm server	Database port

### SCADA Web Server/Web Client Configuration

Default Port	Server Type	Server Role
80	Web server	Project files for web client
2084	Reports server	Reports server communications
2080	Alarm server	Alarm server communications
2085	Trends server	Trends server communications
2078	I/O server	Legacy I/O communications (version 6 or earlier)
2080	Alarm server	Alarm properties connector
2082	I/O server	Publish, subscribe I/O server communications
5482	Alarm server	Database port
5500–5509	Web client/ web server	Range of ports for server advise between web server and web client, for alarm notifications. Inbound on client; outbound on server.

## Launch and Close the Runtime Environment

**Launch Runtime:** To launch the runtime environment: from left side of the PowerSCADA Studiowindow, click Run ( ).

**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:



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 114.

In this section, you will find these topics:

[Log On to the Runtime Environment](#) on page 170

[View the Interface](#) on page 170

[View the Alarms/Events Page](#) on page 172

[Use Security Viewer](#) on page 178

[Security Viewer Filter](#) on page 178

[When Alarms do not Display Correctly](#) on page 174

[Event/Alarm Log Columns Table](#) on page 175

[Alarm/Event Filter Form](#) on page 176

[Use the Analysis Page](#) on page 179

[Use the Equipment Pop-Up Page](#) on page 180

[Perform IEC 61850 Advanced Control](#) on page 185

[View the Tag Viewer](#) on page 184

## Log On to the Runtime Environment

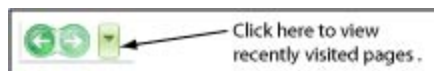
In the upper right-hand corner, click the Login link. Then enter your User ID and password. The features that are available will vary, depending on your user level.

## 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 114](#). These tabs display in rows:


- The upper row (Line 1 in the Menu Configuration file) 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 rows (Lines 1–4) list 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 115](#) for details on customizing genies.

### Lockout/Tagout Icon

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

Lockout/tagout icon: 

**⚠ DANGER**

**HAZARD OF ELECTRIC SHOCK, EXPLOSION, OR ARC FLASH**

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

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

See [Enable Lockout/Tagout on page 113](#) for instructions on enabling the lockout/tagout feature.

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

### Communications Loss

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

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

## View the Alarms/Events Page

See also: [Use the Equipment Pop-Up Page on page 180](#)

To view the alarms or events, click the Alarms/Events tab; then select the tab for the Alarm Log or Event Log. The Event Log displays all alarms and events that have occurred. The Alarms Log displays enabled alarms.

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

### Equipment Column

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

(Show All Alarms)

- High\_Voltage
- BusTies
- Generators
- Incomers
- Transfers
- Low\_Voltage
- BusTies
- Incomers
- Lighting
- Motors
- Office
- Medium\_Voltage
- Memory\_Device
- PLSDCluster\_Ne...

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 176](#) 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 176](#)). From the Advanced Alarm Filter window, you can filter by date range, by text matches for various attributes, or by alarm type.

Notice the alarm colors:

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

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

- acknowledge or disable the alarm
- view alarm detail (similar to the genie status page in the single lines of the runtime environment)
- view waveforms: (If the [equipment name Waveform] option does not display, there are no waveforms for this alarm.) Waveforms can display only if the device is set to "acquire on event," and the waveform option is checked in the Profile Editor (see [Enable Waveforms on page 57](#)).

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

After the selected waveform displays, you can view a PDF file that describes the operation of the waveform viewer. Access this file (WaveWeb.pdf in the PowerSCADA Studio Bin folder (64-bit example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Bin).

Waveforms must be correctly set up before they will display. See [Enable Waveforms for Onboard Alarms on page 156](#) for more information. See [Use the Equipment Pop-Up Page on page 1](#) 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.

### Acknowledge, Silence and Print

Each of the logs includes these buttons:

Button	Description
	<p>Acknowledge Current Page of Alarms: Click to acknowledge all of the alarms that display on the current page.</p> <p><b>Note:</b> 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."</p>
	<p>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.</p>
	<p>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:</p> <p>Windows XP: Documents and Settings\All Users\Application Data\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Data</p> <p>Vista operating systems or later: ProgramData\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Data</p> <p><b>Notes:</b> When printing: To avoid truncation of data, choose the Landscape orientation.</p> <p>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).</p>

### 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.	<p>This is due to the difference between way the two alarm types are handled:</p> <p>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.</p> <p>For the PC-Based alarm, the condition for this is, for example, IA &gt; 80; if the tag value for IA is &gt; 80, the appearance will show. The tag is constantly scanned, so the condition triggers the alarm once it is enabled.</p> <p>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.</p> <p>Thus, there is no resolution.</p>	
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.	<p>Use one or more of these procedures:</p> <p>Set alarm filtering in the alarm viewer to reduce the number of alarms that can display.</p> <p>Only support a one-cluster system.</p> <p>If a multiple-cluster system is necessary, display a separate alarm page for each cluster.</p>
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 <a href="#">Alarm/Event Filter Form on page 176_</a>
Page Down button causes an empty page to display.	The last alarm was on the previous page. When there are no more alarms, pressing Page Down displays a blank page.	Click Page Up to return to the previous page (and the last alarms for the system).

## Event/Alarm Log Columns Table

This table lists the column headings that are available for use when viewing the event log and alarm logs. 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

Column Name	Description
Category	event; or high, medium, or low alarm
Change	alarm logs only: when the alarm changes state: first state, second state
Cluster	cluster name to which the alarm belongs
Comment	alarm log only: displays comments from the alarm
Custom3 through 8	custom filters
Date	date (MMDDYYYY) that the event occurred or that the alarm annunciated
Description	description of alarm e.g., Sag Vcn or Under Voltage B-C
Equipment	default equipment name displays; used for alarm filtering and viewing
Help	help page
Location	Onboard or PC-Based
LogState	alarm logs only: The last state that the alarm passed through.
Millisec	alarm logs only: time (MS) that alarm annunciated
Operator	user name from the PowerSCADA Studio users list
Priority	the alarm category's priority
Priv	privilege = security level
State	event log: state of the entry in the event log.. event alarm log: disappearance, appearance
Tag	alarm tag
Time	time (HH:MM:SS:MS) that alarm annunciated
Time Quality	This column displays the quality (accuracy) of the time stamp for alarms/events. Use the "Time Sync" filter to display only data that has confirmed time quality in the log (see <a href="#">Alarm/Event Filter Form on page 176</a> for instructions on enabling the filter). When there is no SER data, this column reads "No Time Sync Information." When the filter is set to Yes, the view displays only the available time sync information from SER devices. Note: If there is no SER data from any device and the filter is set to Yes, the entire log will be blank.

## Alarm/Event Filter Form

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

[Alarm] UsePLSFilter

default: 1 (use PLSCADA filter form)

Change to 0 to use the Citect filter form.

To filter for the information that displays in the alarm logs and the event log, click Filter (in the upper left corner of the screen). The Advanced Alarm Filter screen displays. The table below describes its settings.

Filter Option	Description: Display all alarms for:
Basic Filter box:	

Filter Option	Description: Display all alarms for:
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 I/O Device Manager; (listed in PowerSCADA Studio > System Model > Equipment > Name)
Cluster	a single cluster, which was added when setting up the project (listed in the PowerSCADA Studio)
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 add or change custom filter assignments, use the AlarmFormat parameter (PowerSCADA Studio > Settings tab > 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. 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 <code>_ PLS_ALM_EVENT</code> (category 1004).
Priority	This is the priority of the alarm category; not used in the default <code>PLS_Include</code> project. As with the category, priority has defaults (see Categories and Priorities table below). You can change these settings in the equipment profiles. <i>However, be sure that you use priority 1 for events.</i>
Time Sync	Yes = in the Alarm or Event Log, only events/alarms with time quality information will be listed. The time sync data displays in the Time Quality column of the log. Data displays to the accuracy recorded at the device. Default: no

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

## Use Security Viewer

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

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

The screen displays a table with these default columns:

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

For more information on these fields, see **Alarm SOE fields** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

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

There are three ways to filter information:

- To the left of the log, you can check one or more of the devices in the system. This filters information to include data only for those devices. When nothing is checked, all devices are included.
- You can turn columns "on" and "off." To add columns, right click in the header area of the log, then choose Insert Column. From the list that drops down, you can check an additional column title. The new column displays to the left of the column you clicked.  
To remove a column, right click on the header of the column you want to delete, then click Remove Column.
- You can filter that data that is included. To do this, use the Security Viewer filter. For instructions on filtering the columns in the log, see [Security Viewer Filter on page 178](#).

## Security Viewer Filter

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

Filter Option	Description: Display all alarms for:
Basic Filter box:	

Filter Option	Description: Display all alarms for:
Start Date/End Date	Choosing only a start date displays alarms from that date to the current date. Choosing only an end date displays alarms for the past year up to that date. For example, to display alarms only for today's date, enter only a start date.
Start Time/End Time	Choosing only a beginning time displays alarms from that time through the end of the day (23:59:59 or 11:59:59 p.m.). Choosing only an ending time from the start of the day (00:00:00 or 12:00:00 a.m.) through the time selected.
Cluster	This is a single cluster, which was added when setting up the project (listed in PowerSCADA Studio, Topology window; click Edit, then choose Clusters from the dropdown link)
Area	Area (Between 0 and 255). See <b>Alarm SOE fields</b> in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).
Classification	The class of the event. See <b>Alarm SOE fields</b> in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).
Operator	The user ID of the person who has logged on PowerSCADA Expert.
Message	This comes from the Message field in the Alarm Log.
Custom Filter	There are eight custom filters, which can be assigned by the customer in each alarm. A group of alarms in a specific location could have the same name in CUSTOM8 so that custom filtering can be easily applied. Custom8 has a default assignment of "Equipment." To add or change custom filter assignments, use the AlarmFormat parameter (PowerSCADA Studio > Setup tab > Parameters). This is the only means available for filtering on a custom field. When viewing the log, you can use the new custom filter by typing it into the Custom Filter field.

## Use the Analysis Page

The Analysis Page offers two options for viewing data trends. In either option, you must select the tag(s) that are to be included. To be available for viewing in trends, a tag must be included in a device profile, and it must have the "Trend Tag" box checked (see for more information).

Trend data is automatically logged when you check "Trend Tag" for tag and then add it to the project. If too many tags are chosen as trend tags, it could cause the hard drive to fill up.

NOTE: The maximum number of tags (pens) that will display correctly on the screen is ten. If you exceed ten pens, labels for these pens will not display correctly. Use one of these methods to correct this issue:

1. Enlarge the window to accommodate the extra pens/labels.
2. Write custom code to cause the labels to always be in the same position, overlapping each other when the trend pen is created. The user can then move the label around for better viewing.
3. As with option number two, control the label positions with code; but then, move the labels back to that same spot when a user selects the trend pen again.

There are two methods of calculating disk space usage: scaled and floating point. For more information on these calculations, see **Calculating Disk Storage** in the citectSCADA.chm help file.

- 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 in the trend tags, or until it is FIFO'd out. Some tags are polled every 5 seconds and are stored for two weeks. These tags are:

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

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



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

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

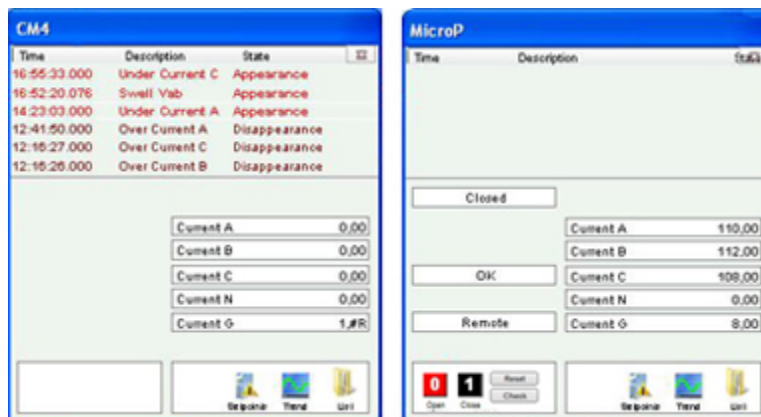


## Use the Equipment Pop-Up Page

See also: [View the Alarms/Events Page on page 172](#)

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 237](#) for user access levels).

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





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

- a waveform. (If you do not see “Waveform” in the list when you right-click the alarm, there are no waveforms for this alarm.) Waveforms can display only if the device is set to “acquire on event,” and the waveform option is checked in the Profile Editor (see [Enable Waveforms on page 57](#)).

When the waveform is available for viewing, it displays when you click this link. For information about how the waveform viewer works, see the WaveWeb.pdf file in the PowerSCADA Studio bin folder (64-bit example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\v8.2\bin).

Waveforms must be correctly set up before they will display. If there are multiple waveforms, you must select from the list that displays (by default, the waveform search returns all waveform files acquired within the 24 hours prior to the time of the alarm). See [Enable Waveforms for Onboard Alarms on page 156](#) for more information. See [Use the Equipment Pop-Up Page on page 1](#) 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 MMXU1A\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.

## Perform IEC61850 Advanced Control

To begin using the advanced control feature, click Check in the lower left section of the window. See [Set Up IEC 61850 Advanced Control on page 116](#) for information on setup. See [Perform IEC 61850 Advanced Control on page 185](#) for information on performing this advanced control.

## View Waveforms

After you select a waveform for viewing from the genie status page, the external waveform viewer displays it. For instructions on using the tool's analysis feature, see WaveWeb.PDF, located in the Bin folder of the PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards Bin folder (example: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Bin).

## Enter Setpoints for Alarms

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

To add setpoints for alarms, follow these steps:

1. View the page, then click the genie for which you want to enter setpoints. A status window displays with the name of the genie.
2. Click Setpoints, then choose Analog, Digital, or All. When the Alarm Setpoints screen displays, select the first value you want to change. At the “keypad” screen (see below), enter the new value. Click OK to save it. Do this for each setpoint that you want to change.



Based on these setpoints, alarms can begin to display both in the alarms window at the top of the runtime screen and on the Alarms/Events tab (assuming you have set one up for this installation).

When there is a comms loss for a device: the last state before the loss happened is displayed on the screen.

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

## View Real-Time Trends

This option displays an historical trend. The data that displays is determined by the value that was selected in the Value Type field when this genie was added to the one-line page.

To view a trend: From the one-line page in the runtime environment, click a genie to view its status window. Click Trend, in the lower right corner. The Analyst screen displays for that trend.

You can select the timeframe for the trend. You can also uncheck phases to remove them from the trend, or highlight a phase to bring it to the front of the trend. For detailed information about the buttons on the screen, click "?" at the top of the page.

### View Lists of Real-Time Information for the Genie

To view lists of real-time currents, voltages; powers; resets and controls; and miscellaneous readings, click List, in the lower right corner, then click an item from the list: Currents, Voltages, Powers, Resets, or Others.

For resets and controls, which are interactive, you should assign users a high level of security. For a list of the default user levels, see [Add and Modify User Accounts on page 237](#). That link also includes information on creating unique users.

When you click an item from the list, individual tag readings display for that tag type (depending on the tags that you have chosen for this device type). When you click any item in that list, the tag pop-up menu displays with these options: Trend, Override Tag, Control Inhibit Tag, and Tag Status. **See Override Tag Status**, below, for details.

### Override Tag Status

From the list, you can right-click individual tags and override status settings. To access this feature, the user account must be at least level 4.

**Trend:** This link allows you to view a trend for the tag that you clicked.

**Override Tag:** You can use this feature to override a real-time value that is incorrect, or to test graphics. Enter the value that you want the system to "read" for this tag in the Override Value line. When you click Apply, the tag is highlighted. When you have finished the test, return to this list to remove the override.

**Control Inhibit Tag:** When this feature is ON, you will not be able to process writes for this tag. To enable this inhibit, click Apply for this tag from the list. The tag reading is highlighted. To disable this feature, return to the list view of this tag; click Remove.

You can perform control inhibit on an entire device. To do this, you will use the IODeviceControl Cicode function. For more information, see the **I/O Device Properties** topic in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

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

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

To change the color for tag overrides, use OverrideTextBackgroundColor. To change the color for control inhibits, use ControlInhibitTextBackgroundColor. For detailed help, see Page Parameters in the Parameters.chm help file (Start > Programs > Schneider Electric > PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards > PowerSCADA Expert Web-based Help).

## View the Tag Viewer

Use the tag viewer to learn the status of all of your project tags. This can provide information that you need to troubleshoot the project while you are commissioning it.

You can filter the tags that you view by individual equipment included in the project. You can also filter on strings that are part of the tag description or tag name. The tag viewer will work in all supported screen resolutions.

To view tags:

Click the tab for the page that was used when setting up the tag viewer, then select the tag viewer. The viewer displays in a screen similar to this:

Tag Description	Tag Name	Value	Timestamp	Quality
Demand Reactive Power KVAR - Pres	PLDCluster PM870_TPRS_MSTA1AAVAr	0.00 kVAR	2015-11-13 09:14:49	Good
Displacement Power Factor	PLDCluster PM870_TBRXU1ZTOPDap	-0.93	2015-11-13 09:14:55	Good
Displacement Power Factor A	PLDCluster PM870_TBRXU1ZDFDapA	-0.91	2015-11-13 09:14:55	Good
Displacement Power Factor B	PLDCluster PM870_TBRXU1ZDFDapB	-0.95	2015-11-13 09:14:53	Good
Displacement Power Factor C	PLDCluster PM870_TBRXU1ZDFDapC	-0.93	2015-11-13 09:14:55	Good
Max Active Power	PLDCluster PM870_TGLR_MSTA1MaxVAr	1.00 kW	2015-11-13 09:14:49	Good
Max Apparent Power	PLDCluster PM870_TGLR_MSTA1MaxVA	1.00 kVA	2015-11-13 09:14:49	Good
Max Reactive Power	PLDCluster PM870_TGLR_MSTA1MaxVAr	0.00 kVAR	2015-11-13 09:14:49	Good
Min Active Power	PLDCluster PM870_TGLR_MSTA1MinVAr	1.00 kW	2015-11-13 09:14:49	Good
Min Apparent Power	PLDCluster PM870_TGLR_MSTA1MinVA	1.00 kVA	2015-11-13 09:14:49	Good
Min Reactive Power	PLDCluster PM870_TGLR_MSTA1MinVAr	0.00 kVAR	2015-11-13 09:14:49	Good
Peak Demand Apparent Power	PLDCluster PM870_TWSTA1MaxVA	1.00 kVA	2015-11-13 09:14:49	Good
Peak Demand Reactive Power	PLDCluster PM870_TWSTA1MaxVAr	0.00 kVAR	2015-11-13 09:14:49	Good
Power Demand Interval	PLDCluster PM870_TWSTA1ZVOMdInt	15.00 Min	2015-11-13 09:14:49	Good
Power Demand Sub-Interval	PLDCluster PM870_TWSTA1ZVOMdSubInt	1.00 Min	2015-11-13 09:14:49	Good
Power Factor	PLDCluster PM870_TBRXU1TOPF	-0.94	2015-11-13 09:14:49	Good
Power Factor A	PLDCluster PM870_TBRXU1TOPFA	-0.90	2015-11-13 09:14:55	Good
Power Factor B	PLDCluster PM870_TBRXU1TOPFB	-0.87	2015-11-13 09:14:49	Good
Power Factor C	PLDCluster PM870_TBRXU1TOPFC	-0.85	2015-11-13 09:14:49	Good
Reactive Power	PLDCluster PM870_TBRXU1TOPR	0.00 kVAR	2015-11-13 09:14:49	Good
Reactive Power A	PLDCluster PM870_TBRXU1TOPRA	0.00 kVAR	2015-11-13 09:14:49	Good
Reactive Power B	PLDCluster PM870_TBRXU1TOPRB	0.00 kVAR	2015-11-13 09:14:49	Good
Reactive Power C	PLDCluster PM870_TBRXU1TOPRC	0.00 kVAR	2015-11-13 09:14:49	Good
EN60180 - Temporary power (Tag Over)V Ex	PLDCluster PM870_TPRD1ZDagierDirVdIntg	0	2015-11-13 09:14:49	Bad
Lagging Power Factor Total	PLDCluster PM870_TPR_DPRV1ZLpSndIntg	0	2015-11-13 09:14:49	Bad
Over Power Apparent Power Total	PLDCluster PM870_TQVA_DPRV1ZQVPrDirVdIntg	0	2015-11-13 09:14:49	Bad

Note the following features:

- **Filter by equipment:** The left-hand pane gives you the option to filter by equipment name. Most equipment is grouped by voltage level. You can select one equipment node, and you will view the tags for that equipment.
- **Filter by tag:** In the upper right corner of the screen, type the tag name. You can type a string, such as "power factor," and you will retrieve a list of tags that have "power factor" in their tag description or tag name.

NOTE: Any time you display a tag, you add to the dynamic point count. See "Dynamic-point Count Licensing" in the citectscada.chm help file (default location: Program Files > Schneider Electric >PowerSCADA Expert > v8.2 > bin) for more information about point counts.

The viewer includes the following columns:

- **Tag Description/Tag Name:** the description and name used when the equipment was added to PowerSCADA Expert.
- **Value/Timestamp:** The real-time value that was read at the date/time shown.
- **Quality:** The data quality (for example, Good or Bad) of the tag from PowerSCADA Studio.

Use the Previous and Next links at the bottom right of the page to scroll through multiple pages.

## Perform IEC 61850 Advanced Control

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

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

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

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

### WARNING

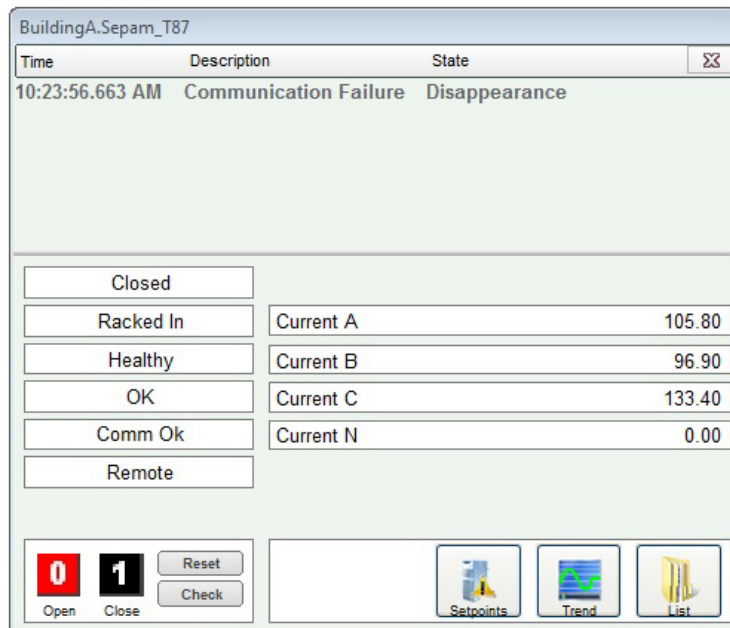
#### UNINTENDED EQUIPMENT OPERATION

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

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

NOTE: For information on enabling this feature, see *Set Up IEC 61850 Advanced Control* on page 116

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



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

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

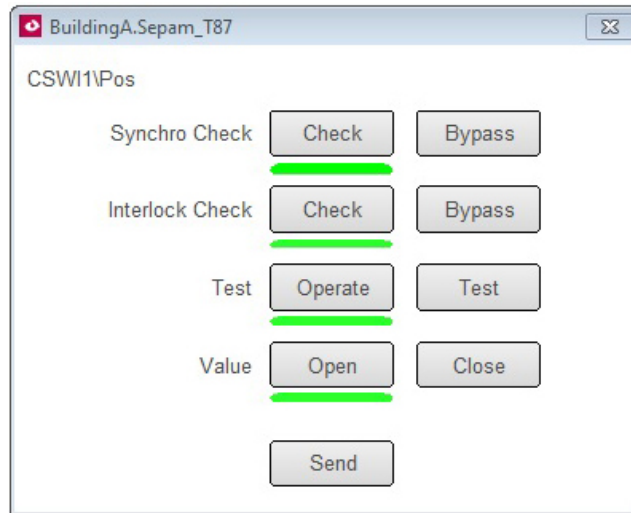
  

0	1	Reset
Open	Close	Check

Setpoints	Trend	List
-----------	-------	------

Click Check, and the advanced control window opens:



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

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

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

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

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

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

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

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

## EcoStruxure WebServices (EWS)

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

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

EWS is set up and configured using the Application Configuration Utility (Start > All Programs > Schneider Electric > PowerSCADA Expert 8.2 with Advanced Reporting

and Dashboards > Config Tools > Application Config Utility. For details, see [EcoStruxure Web Services Setup on page 222](#).

## Use Basic Reports

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

Before you can set up Basic Reports and generate/view reports, you must set up data acquisition parameters. To do this, use the Application Configuration Utility. See [Application Services Host—Citect Data Platform on page 223](#) and (for receiving reports via email) [Basic Reports on page 225](#) for instructions.

In the PowerSCADA Studio, Visualization page, use the Menu Configuration page to configure menu tabs to use the new "PLS\_ReportDsp()" Cicode function to send URLs to the web browser control at runtime. The control then browses to the available reporting web pages. See the PLS\_Example project for examples of this functionality.

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

To get started setting up a report, see [Set up the Runtime Environment for Basic Reports on page 187](#).

For descriptions of each report type, see [PowerSCADA Expert Basic Reports on page 195](#).

**NOTE:** If you install Matrikon Explorer on the same computer as PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, the LiveView and reporting features will not launch. To prevent this, install Matrikon before you install PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. If you install Matrikon after you install PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, you need fix the issue in this way: Go to IIS > ISAPI Filters, and then reset the dll that is already selected (click browse and re-select *v4.0.30319 aspnet\_filter.dll*). Click OK.).

## Set up the Runtime Environment for Basic Reports

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

### Create the Menu Items for Report Page

For a complete discussion of reporting web application URLs, see [Create and View Basic Reports on page 189](#).

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

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

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

**Examples:**

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

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

or

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

which opens an unconfigured single device usage report at the parameters entry page

**Add the Following INI Parameters**

To allow trend queries that yield the desired amount of historical data:

```
[Trend]MaxRequestLength =100000000,  
allowable range: 1–100000000  
(example: a value of 70080 would yield two years of data for one device/one topic,  
assuming 15-minute trends)
```

To allow CtAPI to connect remotely:

```
[CtAPI]Remote = 1
```

To define a privilege level for users to view reports:

```
[Reporting]PrivLevel - Default = 0
```

To define an area for users to view reports:

```
[Reporting]Area - Default = 0
```

**Set Up a Display Client for Basic Report Viewing**

To be able to properly interact with the basic reporting web application at a display client, you must set a registry key to force the Microsoft Web Browser ActiveX control to use Internet Explorer 9 emulation.

Create the following DWORD value at the following registry key path:

```
Path: HKEY_LOCAL_MACHINE\SOFTWAREWow6432Node\Microsoft\Internet  
Explorer\MAIN\FeatureControl\FEATURE_BROWSER_EMULATION
```

Value Name: Citect32.exe

Value: 9999 (decimal)

**NOTE:** This registry setting affects the Citect32.exe process only. It has no effect on other applications that use the MS Web Browser ActiveX control.

**Tasks in the Basic Reporting Application**

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

[Perform IEC 61850 Advanced Control on page 185](#)

[Create and View Basic Reports on page 189](#)

[Configure Email Settings to Send Basic Reports on page 191](#)



[Send Basic Reports via Email on page 192](#)

[Read, Export, Print, and Edit Basic Reports on page 194](#)

## Create and View Basic Reports

This section explains how you can create basic reports and save report configurations using a web browser (e.g., Internet Explorer).

For instructions on interacting with the reporting web application via the runtime environment, see [Set up the Runtime Environment for Basic Reports on page 187](#).

You can generate basic reports in two ways:

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

## Run a New Basic Report

There are two options for this:

1. Browse directly to the URL of the report. Use this format:

`http://<ServerName>/Reporting/Report/<ReportName>`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

Standard report names are:

- MultiDeviceReport
- SingleDeviceReport
- TabularReport
- TrendReport

2. Browse to the default reporting URL, and click the report you want to view.

An example of this format is:

`http://<ServerName>/Reporting/`

where:

<ServerName> = the name or IP of the reporting server

Additional formats are required, depending on the operating systems. For more information, see [URL Routing for Basic Reports on page 199](#).

## Run a Basic Report and Save its Configuration

**NOTE:** If you plan to view a basic report using [Rapid Access Labels on page 217](#), you must save a configuration. After it is saved and you generate a rapid access label, do not change the configuration name. If the configuration name is changed, you must generate a new rapid access label.

To create and save a basic report configuration so you can view it later:

1. Visit the "build configuration" URL of the report you wish to use. Use this format:

`http://<ServerName>/Reporting/Report/<ReportName>/BuildConfiguration`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

2. Enter the report query parameters.

**NOTE:** If you enter a fixed date range, all reports that you generate with this

configuration will use that date range. The best practice is to use one of the relative date ranges, such as "last month."

3. After the report displays, scroll to the bottom of the page. You should see a text box with XML. This XML represents your saved report configuration. Copy the entire contents of the text box into the text editor of your choice.
4. Save this new file to the Reporting\ReportConfigurations\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

```
C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA  
Expert Reporting\Reporting\ReportConfigurations\
```

The filename **must** be in the following format:

```
<ReportName>_<ConfigurationName>.cfg
```

where:

<ReportName> = the name of the report (such as MultiDeviceReport, SingleDeviceReport)

<ConfigurationName> = a name for this configuration (alphanumeric only)

If you use Notepad, ensure that you apply the correct suffix (.cfg) and not the default (.txt).

## View a Basic Report Using a Saved Configuration

To view a basic report with a saved configuration, browse to the URL of the report and specify the configuration. This method runs the report directly with the saved configuration (you cannot change the parameters).

This URL is in the following format:

```
http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>
```

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

<ReportConfiguration> = the name of the saved configuration to use

## Modify and View a Basic Report Using a Saved Configuration

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

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

```
http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>/ShowC  
onfiguration
```

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you want to view/modify

<ReportConfiguration> = the name of the saved configuration to use

Run the report as you normally would, editing selections on the parameter entry page as necessary.

After the report runs, a text box displays at the bottom of the finished report. It contains the new XML of your saved configuration.

Copy and paste this new XML into your saved configuration file (overwriting the old XML).

## Remove a Saved Configuration

To remove a saved configuration, delete the saved configuration file from Reporting\ReportConfigurations\ directory, described above:

Example (64 bit):

```
C:\Program Files (x86)\Schneider Electric\PowerSCADA  
Expert\v8.2\Applications\Reporting\ReportConfigurations
```

## Configure Email Settings to Send Basic Reports

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

**NOTE:** You must configure the SMTP server and email list(s) before you email reports. See [Send Basic Reports via Email on page 192](#) for instructions on sending these emails.

### SMTP Server and From Address

For instructions on setting up the SMTP server, see [Basic Reports on page 225](#).

### Email Lists

Before you can send email via the URL or ReportMailer method, you must create at least one email list:

1. Open your preferred text editor; enter one or more email addresses (one per line, no commas).
2. Save this text file in the Reporting\ReportConfigurations\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

```
C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA  
Expert\Reporting\Reporting\ReportConfigurations\
```

The filename must be in the following format:

```
Email_<EmailListName>.cfg
```

where:

<EmailListName> = an alphanumeric (no spaces) name for the email list (for example, Administration)

### Email Body

The email body that you send is contained in a resource (.resx) file in the Reporting\bin\Resources\Reporting.en-US.resx\ directory, located on the application root install directory (which is also the physical directory behind the reporting web application's virtual path in IIS).

Example (64 bit):

```
C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\PowerSCADA  
Expert\Reporting\bin\Resources\Reporting.en-US.resx\
```

The email body is the same for all Report Configurations and Email Lists, but you can modify the entry for ReportEmailBody to change the body of the email that is sent.


## Send Basic Reports via Email

Before you can send PowerSCADA Expert basic reports via email, you must configure the SMTP server and email list(s) before you email reports. See [Configure Email Settings to Send Basic Reports on page 191](#) for instructions on configuring this information.

There are three ways to send basic reports via email:

### 1. The Report Viewer email button

Use this method to send a customized one-time email to an individual or group of email addresses.

1. Run the report as normal.
2. Click the Email icon (  ) in the report viewer.
3. Enter the requested information in the pop-up dialog.
4. Click Send.

### 2. Visit a Specific URL

**NOTE:** Each visit to a URL causes the email to be sent. Be sure that you have the correct report and email list before you visit this URL/send the email. Also, you should secure this URL using the web.config file. For information on modifying/using the web.config file, see <http://support.microsoft.com>, and search on kb 815179.

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

`http://<ServerName>/Reporting/Report/<ReportName>/<ReportConfiguration>/Email/<EmailList>`

where:

<ServerName> = the name or IP of the reporting server

<ReportName> = the name of the report you wish to view

<ReportConfiguration> = the name of the saved configuration to use

<EmailList> = the name of the email list you wish to use

You must use a saved configuration (see [Create and View Basic Reports on page 189](#) for instructions). You cannot change report parameters from this URL.

No progress bar or update will display, as these interfere with some scheduling clients.

### 3. Use Cicode via ReportMailer

You can use a utility called ReportMailer to email basic reports. This command line utility is located in the PLS\_Include project. It can be called by Cicode. You can create a button on the graphics page and have it call the Cicode function or use a scheduled process to trigger an email.

Before you can use ReportMailer, you need to create or edit the file called *ReportMailer.ini* file that is in your project (not in PLS\_Include). The ReportMailer.ini file must include the text listed in the table below:

Text Field	Required Setting	Description
LoginUsername	demo	Username for logging in to reporting system for emailing reports
LoginPassword	demo	User's password, will be encrypted on the first run
IsEncrypted	False	Flag that indicates if the password is encrypted. If you change the password, edit the field (replacing the unreadable encrypted entry, if one exists). Then change this value to False. The new password will be encrypted at the next startup cycle, and this field will be updated to True.
ScadaBinPath	C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\Bin	The bin path of PowerSCADA Expert
LogOnUrl	http://SCADASERVER/Reporting/LogOn.aspx	The URL of the logon page(this is an example; use your own server name)
ReportServerName	SCADASERVER	The name or IP address of the server running the reporting application
LogLevel	All	The level of logging you want in the report mailer application. This log is saved to a ReportMailerLog.txt file in the running project's directory. Possible settings are ALL, DEBUG, ERROR, WARN.

After this file is configured, run the ReportMailer.exe with the following syntax:

ReportMailer.exe <ReportName> <ReportConfiguration> <EmailList>  
<ScadaProjectPath>

where:

<ReportName> = the name of the report you wish to view

<ReportConfiguration> = the name of the saved configuration to use

<EmailList> = the name of the email list you wish to use

<ScadaProjectPath> = the full path to your SCADA project

This command line application may be called from Cicode following the below example:

**FUNCTION**

```

PLS_EmailReport()
  ErrSet(1);
  STRING FilePath = ParameterGet("CtEdit","User","") + "PLS_
  Include\ReportMailer.exe " +
  "MultiDeviceReport SampleConfiguration SampleList " +
  "^"C:\ProgramData\Schneider Electric\PowerSCADA Expert 8.2 with
  Advanced Reporting and Dashboards\User\PLS_Example^";
  Exec(FilePath);
END

```

**NOTES**

- The SCADA project path must be enclosed in escaped quotes (^).

- This is an asynchronous (non-blocking) call. While the EXEC() method will return immediately, it may take a few moments to run and email the report. See the web.config timeout value (see option 2 above) for more information.
- You can also call the ReportMailer application directly from a command line. In this case, you can add the term "blocking" to the command line (as a fifth parameter). This causes ReportMailer to act in a synchronous state (block the call) and to return any error messages to the console. Never use the "blocking" parameter by Cicode, as it could prevent EXEC() from returning in a timely fashion.

## Scheduling Basic Reports

You can schedule the emailing of basic reports by executing the above Cicode as an action from a timed event. For more information, see **Configuring Events** in the CitectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

You can also use the Windows Task Scheduler to send these reports. Refer to Microsoft's documentation on the task scheduler at: <http://windows.microsoft.com/en-US/windows7/schedule-a-task>.

### Read, Export, Print, and Edit Basic Reports

After you create a basic report, you can:


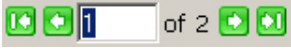


- change its appearance (Page Setup)
- print it
- change its view: HTML or PDF
- scroll to the beginning, previous page, next page, or last page
- export it to a variety of formats
- email it

The toolbar options table below describes these options.

## Toolbar Options

The report toolbar includes several options:

Option	Description
Parameters/Report	Toggle between viewing the parameters (setup) page and the report.
Parent/Child reports	Not currently used.
Hide/Show	Not currently used.
Page Setup	Click this link to open the Page setup window, where you can determine paper size, and page orientation and margins.
Print	Click this link to print the report. NOTE: For best formatting of the report, you should export to PDF, and then print.
Print Preview	In HTML mode, click this link to view the print output.

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 <a href="#">Send Basic Reports via Email on page 192</a> .


## Export a Basic Report

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

You can export to:

- Acrobat (Pdf) File
- Web Archive
- Word Document
- Xml File
- XLS Document

To export:

1. While viewing the report, select a format, then click Export (  ).
2. Type the location at which you want to save the file.
3. Set any other properties you wish.
4. Click Export.

## Edit the Basic Report Appearance

With the report displayed, you can:

- Change the paper size
- Change the paper source
- Change the page orientation
- Change the page margins
- Change the number of pages per sheet
- Add a watermark

## PowerSCADA Expert Basic Reports

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

Before you can view basic reports, there must be data logged for the project. See [Use the Analysis Page on page 179](#) for help.

You must also create a report tab for the graphics page in runtime mode. See [Set up the Runtime Environment for Basic Reports on page 187](#).

**NOTE:** If the Schneider Electric CoreServiceHost has not been refreshed after devices or topics have been added, you should clear the cache and refresh the platform in order to access the new devices or topics.

See [Clear Cache and Refresh Platform on page 161](#) for instructions.

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

[Single Device Usage Reports on page 196](#)

[Multi Device Usage Reports on page 197](#)

[Tabular Reports on page 197](#)

[Trend Reports on page 198](#)

[Rapid Access Labels on page 217](#)

See also:

[Localization for Basic Reports on page 200](#)

## Single Device Usage Reports

Use single device usage reports to display historical energy data from a single device and multiple topics. A single device report includes only usage and consumption topics.

**NOTE:** The report is optimized for up to five topics. If you choose too many topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See [Set up the Runtime Environment for Basic Reports](#).

To set up a single device usage report, follow these steps:

1. Browse to the single device report in the reporting web application. When prompted, enter your PowerSCADA Studio user account information. Click Login.
2. At the next screen:
  - Type a report title.
  - In Reporting Period, choose the date range for this report, for example, *last week*.  
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
  - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)
  - Highlight the name of the device that you want for the report.
  - Check the topics to be included.
3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a usage summary table, and a graph and table for each topic you selected. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see [Read, Export, Print, and Edit Basic Reports](#).



## Multi Device Usage Reports

Use multi device usage reports to display historical energy data for multiple devices and one topic. A multi device usage report includes only usage and consumption topics.

**NOTE:** If you choose too many devices, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a multi device usage report, follow these steps:

1. Browse to the multi device usage report in the reporting web application. When prompted, enter your PowerSCADA Studio user account information. Click Login.
2. At the next screen, do the following:
  - Type a report title.
  - In Reporting Period, choose the date range for this report, for example, *last week*.  
  
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
  - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)
  - Check the names of the devices for the report.
  - Highlight the topic to be included.
3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a usage summary, a value table by interval for all of the devices selected, and a pie chart. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

## Tabular Reports

Use tabular reports to display a system's historical data in a table format. Tabular reports can include one or more devices and one or more topics. A tabular report can include all available topics.

**NOTE:** The report is optimized for up to five topics. If you choose too many devices or topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a tabular report, follow these steps:

1. Browse to the tabular report in the reporting web application. When prompted, enter your PowerSCADA Studio user account information. Click Login.
2. At the next screen, do the following:
  - Type a report title.
  - In Reporting Period, choose the date range for this report, for example, *last week*.  
  
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)

- From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)  
If you leave the default *By Interval*, you will get every data point in the selected date range.
  - Check the name(s) of the device(s) for the report.
  - Check the topic(s) to be included.
3. Click Generate Report.

After the report is generated, it displays as a table on the screen. It lists data for all of the tags according to their timestamps. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

### Trend Reports

Use trend reports to display a system's historical data in a trend (line) and table formats. Trend reports can include one or more devices and one or more topics. A trend report can include all available topics.

**NOTE:** The report is optimized for up to five topics. If you choose too many devices or topics, the chart legend can become unreadable.

Before you can create and view a report, you must set up reporting in the runtime environment. See *Set up the Runtime Environment for Basic Reports*.

To set up a trend report, follow these steps:

1. Browse to the trend report in the reporting web application. When prompted, enter your PowerSCADA Studio user account information. Click Login.
2. At the next screen, do the following:
  - Type a report title.
  - In Reporting Period, choose the date range for this report, for example, *last week*.  
If you choose *Custom...*, the *Start Date/Time* and *End Date/Time* fields display. Enter the date and hour:minutes:AM/PM. (The date/time fields do not apply for the other reporting periods.)
  - From the Period Grouping dropdown box, choose the interval by which you want to see the data reported. (The options here vary, depending on the date range selected.)  
If you leave the default *By Interval*, you will get every data point in the selected date range.
  - Click the name(s) of the device(s) for the report.
  - Click the topic(s) to be included.
3. Click Generate Report.

After the report is generated, it displays on the screen. It includes a trend for each topic included (selected data points over the period of the trend) followed by a table with every timestamp in the period selected. You will probably have to page forward in the report to see all of the information.

For information about reading, exporting, printing, or editing reports, see *Read, Export, Print, and Edit Basic Reports*.

## Enable Windows Authentication for Basic Reporting

You may want to use Windows Authentication for logging in to the Basic Reports application. This could be to authenticate from Active Directory or to provide a single-

sign-on. Follow the steps below to configure Windows Authentication.

**NOTE:** These steps are specific to Windows 7; they may be different for other operating systems. For further assistance, view Microsoft's documentation on this topic at:

[http://technet.microsoft.com/en-us/library/cc754628\(v=ws.10\).aspx](http://technet.microsoft.com/en-us/library/cc754628(v=ws.10).aspx)

1. Turn on the Windows Authentication feature:
  - a. From the control panel, click Programs and Features > Turn Windows features on or off.
  - b. Check Windows Authentication.
2. Enable Windows Authentication in IIS:
  - a. From the Control Panel > Administrative Tools, choose Internet Information Services (IIS).
  - b. Select the root node from the tree on the left (or the Reporting website, if this server hosts multiple sites).
  - c. From the right hand pane, in the IIS section, click Authentication.
  - d. Enable Windows Authentication.
3. Modify web.config to specify Windows Authentication:
  - a. From the root of the Reporting direction, locate web.config.
  - b. Change the line:

```
<authentication mode="Forms">
to
<authentication mode="Windows">
```

4. Add roles to web.config:

You must add roles to web.config to allow access to the basic reporting application. For example, to allow the role (group) Administrators, add the following to the web.config file:

```
<authorization>
  <allow roles="Administrators"/>
  <deny users="?" />
</authorization>
```

Modifying the web.config file is an advanced topic that is covered on the Microsoft website.

See the following link for instructions provided on the Microsoft website:

<http://www.iis.net/configreference/system.webserver/security/authentication/windowauthentication>

Additional information may be available in the following knowledge base article:

<http://support.microsoft.com>, and search on kb/815179.

## URL Routing for Basic Reports

The basic reporting application uses Asp.NET extensionless URL routing. Depending on your operating system, you may need to follow additional steps to enable URL routing in your project.

### Windows 2008 R2 and Windows 7

Microsoft has discovered an issue with extensionless URL routing in certain installations of IIS 7.0 and IIS 7.5. To address this issue, Microsoft has released a hotfix

referenced by KB article 980368. This hotfix is available at <http://support.microsoft.com/kb/980368>.

This hotfix is included in Service Pack 1 for Windows 2008 R2 and Windows 7. To receive the hotfix, you should install Service Pack 1. This installation provides additional important updates to the operating system. To obtain Service Pack 1 for Windows 2008 R2 and Windows 7, go to either Windows Update or <http://support.microsoft.com/kb/976932>.

### Windows XP

When you install IIS on a Windows XP operating system, IIS version 5.1 is installed. When browsing to the default URL, you must append to the URL "default.aspx".

Example:

`http://<servername>/Reporting/default.aspx`

## Localization for Basic Reports

To localize basic reports, you must first rename and translate the .resx files into the target language(s), called "culture names." The .resx files are located in the PowerSCADA Expert Resources directory (default location: C:\Program Files (x86)\Schneider Electric\PowerSCADA Expert\v8.2\Applications\AppServices\bin\Resources).

Make copies of the .resx files listed in the table below. Change the extensions of the copies into the culture name for the target language. Note that you do not need to include both the culture ("es") and the region ("ES"), unless you specifically want a language that is different from the default.

For example, BinningResources.en-US.resx would be named BinningResources.es-ES.resx when you localize it into Castillian Spanish.

For a list of the culture names you can use, see the *National Language Support (NLS) API Reference*, currently at:

[NLS API](#)

After you change the extensions, provide translations for the strings in each .resx file. Store the localized .resx files in the same Resources folder as the English versions.

## Template Editor

Use the Template Editor to begin creating, duplicating, modifying, and deleting LiveView templates and views.

See [Set Up LiveView on page 209](#) for complete instructions.

## Use LiveView

PowerSCADA Expert provides the ability to create and view LiveView templates and views. Some basic predefined templates are included with the software; you can create new templates or make copies of the predefined templates and edit the copies.

Before you view LiveView templates and views, you must set up data acquisition parameters. To do this, use the Application Configuration Utility. See [Application Services Host—Citect Data Platform on page 223](#) for instructions.

### NOTES:

- If you find that a predefined table does not include enough cells for the data you want to display, use the duplicate feature to make a copy of the predefined table. Then add the needed cells to the duplicate.
- If you install Matrikon Explorer on the same computer as PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, the LiveView and reporting features will not launch. To prevent this, install Matrikon before you install PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards. If you install Matrikon after you install PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards, you need fix the issue in this way: Go to IIS > ISAPI Filters, and then reset the dll that is already selected (click browse and re-select *v4.0.30319 aspnet\_filter.dll*). Click OK.).

You can only view data in these templates if your system is online and you are connected to devices that provide data.

To begin creating LiveView templates, click Start > Programs > Schneider Electric > PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards > Config Tools > Table Editor. The LiveView Template Editor screen displays.

To view a template, see [LiveView Viewer on page 207](#).

To create a new template, see [Set Up LiveView on page 209](#).

**See also:**

[LiveView Formulas on page 213](#)

[LiveView Placeholders](#)

[LiveView Thresholds on page 214](#)

[LiveView Formatting on page 211](#)

## LiveView Tables

Click any of the following links to learn about the live view tables:

[LiveView Basic Readings Summary on page 201](#)

[LiveView Power Flow Summary on page 202](#)

[LiveView Energy Summary on page 202](#)

[LiveView Energy Readings on page 202](#)

[LiveView Fundamental Phasor Readings on page 203](#)

[LiveView THD Current Summary on page 203](#)

[LiveView THD Voltage Summary on page 203](#)

[LiveView Uptime Summary on page 204](#)

[LiveView Incremental Reactive Energy Summary on page 204](#)

[LiveView Incremental Real Energy Summary on page 204](#)

[LiveView Harmonic Apparent Power Flows on page 204](#)

[LiveView Harmonic Reactive Power Flows on page 205](#)

[LiveView Harmonic Real Power Flows on page 206](#)

[LiveView Demand Current Summary on page 206](#)

[Live View Demand Voltage Summary on page 206](#)

### LiveView Basic Readings Summary

This summary displays real-time basic power information for a selected device or devices. After opening the basic readings summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The basic readings summary provides this data:

- voltage A-B (V)
- current A (A)
- real power (kW)
- power factor

### LiveView Power Flow Summary

This summary displays a power flow summary for your system devices. Use the information from this table to help optimize the system's power flow.

After opening the power flow summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The power flow summary provides this data:

- real power (kW)
- reactive power (kVAR)
- apparent power (kVA)
- demand average (kW)
- demand peak (kW)
- predicted demand (kW)

### LiveView Energy Summary

This summary displays an energy summary for your system devices. Use the information from this table to help monitor the system's energy consumption.

After opening the energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The energy summary provides this data:

- real power (kW)
- reactive power (kVAR)
- apparent power (kVA)
- block demand real power (kW)
- thermal demand real power (kW)
- peak block demand real power (kW)
- peak thermal demand real power (kW)
- block demand real power predicted (kW)
- thermal demand real power predicted (kW)

### LiveView Energy Readings

This table displays accumulated energy readings for a single device. Data is accumulated beginning with the last energy reset for the device.

Energy values, will be according to one of these accumulation methods:

**Absolute** (unsigned): The device stores positive energy values, regardless of the direction of power flow. The energy value increases, even during reverse power flow.

**Signed:** The device stores both positive and negative energy values. The energy value increases or decreases, depending on the direction of the power flow.

After opening the live view energy readings table, choose the device you want and set the update interval for this table. Click Display Table.

The live view energy readings table provides these accumulated readings:

- real energy (kWhr)
- reactive energy (kVARHr)
- apparent energy (kVAHr)

### **LiveView Fundamental Phasor Readings**

This summary displays a fundamental phasor readings table for a single device, to confirm that the system is properly wired.

After opening the fundamental phasor reading template, choose the device for which you want readings and set the update interval for this table. Click Display Table.

The fundamental phasor readings table provides a phasor diagram that indicates current and voltage magnitudes and angles for each phase.

### **LiveView THD Current Summary**

This summary displays a THD current summary for your system devices. Use the information from this table to monitor your equipment and system power quality.

After opening the THD current summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The THD current summary provides this data:

THD current (%):

- phase A
- phase B
- phase C
- neutral

### **LiveView THD Voltage Summary**

This summary displays a THD voltage summary for your system devices. Use the information from this table to monitor your equipment and system power quality.

After opening the THD voltage summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The THD current summary provides this data:

THD voltage (%):

- Vab
- Vbc
- Vca
- Van
- Vbn
- Vcn

### LiveView Uptime Summary

This summary displays an uptime summary for your system devices. Use the information from this table to view the number of seconds the equipment has been in uptime (defined as all three phases > 10% nominal), and to view the percentage of uptime vs. total time. The summary includes the last 12 months.

After opening the uptime summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The uptime summary provides this data:

For the past 12 months:

- Uptime %
- Uptime
- Downtime

### LiveView Incremental Reactive Energy Summary

This summary displays an incremental reactive energy summary for your system devices. Use the information from this table to monitor transmission of energy beyond the previous baseline, to help achieve optimum loading.

After opening the incremental reactive energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The incremental reactive energy summary provides this data:

- incremental reactive energy into the load (kVARHr)
- incremental reactive energy out of the load (kVARHr)
- date/time of the last incremental energy update

### LiveView Incremental Real Energy Summary

This summary displays an incremental real energy summary for your system devices. Use the information from this table to monitor the energy usage and production above the previous baseline, to help achieve optimum loading.

After opening the incremental real energy summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The incremental real energy summary provides this data:

- incremental real energy into the load (kVARHr)
- incremental real energy out of the load (kVARHr)
- date/time of the last incremental energy update

### LiveView Harmonic Apparent Power Flows

The harmonic apparent power flows table displays real-time apparent power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic apparent power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.



You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic apparent power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- remaining hold
- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
  - power flow in from the utility
  - power flow out to the utility
  - apparent power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

### **LiveView Harmonic Reactive Power Flows**

The harmonic reactive power flows table displays real-time reactive power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic reactive power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.

You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic reactive power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- Remaining hold
- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
  - power flow in from the utility
  - power flow out to the utility
  - reactive power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

## LiveView Harmonic Real Power Flows

The harmonic real power flows table displays real-time real power flow information. Use this information to help determine the impact of harmonics on system equipment.

After opening the harmonic real power flows template, select the device, then click Display Table.

In the upper right, you can set the update interval for this table.

You can set meter registers to enable frequency domain analysis of waveforms and the format used in analysis. Harmonics and trend tables reflect these register settings. For details about these settings, read the Notes to the right of the table.

The harmonic real power flows table provides this data:

- meter type
- wiring type
- FFT magnitudes
- FFT enable
- FFT status
- FFT hold time
- remaining hold
- total voltage harmonic distortion for all three phases
- total current harmonic distortion for all three phases
- harmonic components for all three phases:
  - power flow in from the utility
  - power flow out to the utility
  - real power

Magnitudes and angles are available for all odd harmonics from H1 through H31.

## LiveView Demand Current Summary

This summary displays a demand current summary for your system devices. Use the information from this table to help monitor the system's demand current.

After opening the demand current summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The demand current summary provides this data:

average demand current and peak demand (both in amps)

- Ia
- Ib
- Ic

## Live View Demand Voltage Summary

This summary displays a demand voltage summary for your system devices. Use the information from this table to monitor the system's demand voltage.

After opening the demand voltage summary template, you set the update interval for this summary. You can also add or remove devices from the summary.

Make your selections, and click Display Table.

The demand voltage summary provides this data:

demand voltage

- Vab
- Vbc
- Vca
- Van
- Vbn
- Vcn

## LiveView Viewer

Use this screen to view table templates, and to view or create table views, in the LiveView Viewer.

To open this screen, in Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 210](#)). In the PLS\_Example project, there is a tab for LiveView. For information about an individual table, click a link from the Contents folder.

NOTE: If you plan to view a table using the [Rapid Access Labels on page 217](#) feature, do not change its name after you print the QR code. If the name is changed, you must generate a new rapid access label.

### Open LiveView from a URL Link

Before you can open LiveView from a URL link, you must select a template and the desired devices, display the table, and save it as a View.

To open this view using a URL link, use one of these options:


- From the computer where LiveView is installed, enter  
`http://localhost/LiveViewViewer`
- From a remote client computer, enter  
`http://10.10.10.10/LiveViewViewer`  
(where 10.10.10.10 is the URL of the server where LiveView resides)

To automatically open a specific table when you launch LiveView Viewer, add the table name to the end of the address. For example, to open the basic readings summary view while on the local computer, you would enter:

`http://localhost/LiveViewViewer/Basic Readings Summary View`


### LiveView Viewer Display

The Live View Viewer displays with two tabs, Templates and Views.

**Templates:** A template includes all setup data (placeholders, formulas, thresholds, and formatting); but it does not have devices selected. The templates include those that are predefined (designated by the locked symbol: ), as well as those that have been defined in the Setup window.

To view a template, select it from the list. Then, select the device(s) for which you want to display values. (Only devices that have at least one assigned topic from the topic placeholders in this template are available for selection.)

Click Display Table to view the template in the right-hand pane.

To save a template as a view, display the template. Click () on the upper right of the Viewer page. In the View Name window, edit the name, then click OK. The new view is saved in Tables > Views on the server. The view will also display in the left-hand pane of the Views tab.

**Views:** A view is a template that is saved with its device selection(s). The views listed are saved on the server in Tables > Views. Views are available to all users, whether on the server or a client. They also display on the Views tab of the Live View Viewer.

To open a view, highlight its name, then click Display. The view displays in the right-hand pane with updated data. You can delete a view (click Delete, to the right of the View tab). You can change a view by adding or deleting devices and then either overwriting the view or saving it as a new view.

**Update List:** This link forces the cached table and view lists to be refreshed, displaying any newly added tables and views.

**Select Device(s) and Update Device List:** This link forces any new devices (with at least one assigned topic) to display. In the Select Device(s) list, you can move devices higher or lower in the list that you see, so that they display in the order you prefer. To do this, right-click and highlight a device, then click one of these icons:



: Move to the top (double arrow) or move up one step (single arrow)



: Move to the bottom (double arrow) or move down one step (single arrow)

[Where's My Device? on page 209](#): Click this link to explain why an expected device does not display in the table.

### Template and View Features


The template (after you click Display) or view displays with devices and data. The following information is included:

**Placeholders:** All placeholders that were added during setup will display with the appropriate device name or tag value.


**Thresholds:** If any of the tag values are outside of the normal range established in the Thresholds that were added during setup, the font color will reflect the high or low status of that tag.

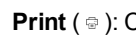
On the right-hand side of the top of the screen are these buttons:




**Save** (  ): Click to save a template as a "view." You are prompted to name the view (default: table name appended with "view"). The view is saved in Tables > Views on the server. The view includes the devices that were selected for the table.



**Notes** (  ): Click to display a description of the table that was added when the table was set up.



**Print** (  ): Click to print a copy of the table with its current values.

**Last Update:** This is the most recent date/time that the template or view values were updated.

**Update Interval:** Choose the interval of time that will pass between requests to update the data in the template or view. Options are:

**Manual:** Updates only occur when you click Update Now.

**5 seconds**

**10 seconds**

**30 seconds**

**1 minute**

**10 minutes**

**Update Now:** Click to manually update values and refresh the template or view.

## Where's My Device?

This help topic displays when you click "Where's My Device?" below the device list in LiveView Viewer.

### Missing Topics

Only devices that have topics available for the selected template will appear in the device selection list. If you do not see an expected device, it is missing because it does not include topics that are used in this template.

If this is a template that you have created, you can open the template in the [LiveView Placeholders on page 212](#) screen of LiveView Template Editor to add the placeholder (s). If this is a predefined template, you cannot change it; you will need to make a duplicate template and then add the desired placeholder(s).

### Clear Cache and Platform Refresh

If the Schneider Electric CoreServiceHost has not been refreshed after devices or topics have been added, you should clear the cache and refresh the platform in order to access the new devices or topics.

See [Clear Cache and Refresh Platform on page 161](#) for instructions.

## Set Up LiveView

Use this application to begin creating, duplicating, modifying, and deleting LiveView templates and views.


You can configure a LiveView template in LiveView Template Editor, and then display it on the server or on a web client. A "template" includes all of the setup data except devices.

You can create "views" of templates in LiveView Viewer. A "view" is a template that includes devices.

To access LiveView Template Editor, click Start > Programs > Schneider Electric > PowerSCADA Expert > v8.2 Config Tools > Template Editor.

Only one user at a time can access LiveView Template Editor. When a user accesses LiveView Template Editor, a file called *TemplateEditor.lock* is saved on the PowerSCADA Expert folder of the server (default location: Program Files > Schneider Electric > PowerSCADA Expert > v8.2 > Applications > LiveView > Viewer). If necessary, an administrator can unlock the utility by deleting *TemplateEditor.lock* from the server.


Here you can see:

- Notes icon (  ): This opens a free-form field to add any descriptive information about the template that will be useful. The information displays in a notes field, to the right of the template. This field closes when you click Done.
- In the left-hand pane are the following:
  - New:** (You are prompted to save if you are editing a template that is not saved.) Click to save the template you are editing, and then to add a new template. The "New Template" name displays in the list, a new template file is uploaded to the server in Table > Templates, and an empty template displays in the right-hand pane. All fields are set to their defaults.

**Duplicate:** Click to save a copy of the selected template. The current template name is used with "Copy" appended. Use this option to edit a predefined template.

NOTE: If you find that a predefined table does not include enough cells for the data you want to display, create a duplicate. Then add the needed cells to the duplicate.

**Delete:** Click to delete the current template (you cannot delete predefined templates). Confirm that you want to delete it. All views associated with the template will also be deleted.

**Select Template:** This list includes all of the templates that are set up. Predefined templates display a lock icon (  ) to the left of the name. These templates cannot be deleted or edited.

**Template Name:** Overwrite the current name, which updates the template here and in the list of templates. This will also update the views that are associated with this template.

**Single Device/Multiple Device:** Click the radio button for the type of template you want (default: Single Device).

**View Area:** Use this field to determine the area of the table that will be viewed in LiveView Viewer. When you set up a table, there may be information (such as formulas or notes) that you do not want to display in the final table in the Viewer. To select only the material that you want to view, do one of the following:

- In the View Area text field, type the cell range that you want to view (for example, A1:D20).
- Select the cells that you want to include, then press Use Selection.

In either case, a border displays around the cells in the range you select.

Save the template. When you view it in LiveView Viewer, it will only include the cells you selected.

**Save:** This button becomes live only after you make a change to a template that is being edited. The template is saved as an .xlsx file; it is uploaded to the server in Table > Templates. The saved template appears in the View tab after you click Save. (You do not need to click Save when you create a new template or a duplicate; these files are automatically saved.)

To create a new template, see [Create a LiveView Template on page 211](#).

To modify an existing template, see [Modify LiveView Template on page 215](#).

To duplicate a template, see [Duplicate LiveView Template on page 215](#).

To delete a template, see [LiveView Delete on page 216](#).

See also:

[LiveView Formulas on page 213](#)

[LiveView Placeholders on page 212](#)

[LiveView Thresholds on page 214](#)

[LiveView Formatting on page 211](#)

## Create Menu Item for LiveView Page

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

1. From the PowerSCADA Studio, click Visualization tab > Menu Configuration.
2. Add the new menu item that you want for each of your LiveView tables.

3. In each of these menu items, in the Menu Command line, add the Cicode method that will display a LiveView tab. You can create your own custom method or use the default:

```
PLS_LiveViewDsp (STRING sIPAddress, STRING sViewName = "", STRING  
sTitle = "")
```

Example:

```
PLS_LiveViewDsp ("10.10.10.10", "BasicReadingsSummary",  
"ShowConfiguration/MyConfiguration", "Basic Readings Summary");
```

which opens a configured LiveView table view with the saved configuration name "MyConfiguration".

## Create a LiveView Template

To add a new LiveView template:

1. Open LiveView Setup, then click New.  
An empty template displays with a "New Template" name.
2. Overwrite the template name with a name of your choice, up to 100 characters (limited to A–Z, a–z, 0–9, spaces, underscores, hyphens, and parentheses).
3. To continue setting up the template, click one of the links below.

**Template Name:** default: New Template (or, if an earlier New Template exists, appended with a 1)

**Single Device/Multiple Devices.** The default is single device. If you want a multiple device table, click Multiple Devices.

NOTE: Table grid lines do not display in the LiveView Viewer, but they do display in LiveView Setup.

See also:

To add data formulas to the real-time table, see [LiveView Formulas on page 213](#).

To add data (device names and tag names) to the real-time table, see [LiveView Placeholders on page 212](#).

To add visual alerts (color changes) when the value of the tag associated with a cell becomes too high or too low, see [LiveView Thresholds on page 214](#).

To add formatting to cells, such as font and font size, see [LiveView Formatting on page 211](#).

## LiveView Formatting

Use this feature to add cell formatting, such as font, font size, and color. To use this feature, click the Formatting subtab in LiveView Template Editor. A formatting toolbar displays on the screen. It allows you to set the appearance of the cells in the template.

NOTE: Formatting changes become visible only after you click outside of the cell that you change.

To format a cell or range of cells, select the cell(s). When you select a format, the active cell(s) will be set to the specified format attribute. When a cell becomes active, the format selections on the toolbar will reflect the selections for that cell. When you select multiple cells, the format selections will reflect those of the first cell you select.

Then you can format the cell appearance:

- Choose font and font size.
- Choose whether to apply bold, italics, or underline.

- Choose a color for the font (default is black), and for the background of the cells (default is white).
- Choose the horizontal alignment: flush left, centered, or flush right.
- Choose the vertical alignment: top, center, or bottom.
- If more than one cell is selected, Merge Cells is enabled. Check this box to merge the selected cells into one large cell.
- In the Data Type drop-down box, select the type of data that will be in the selected cell(s):
  - **Text** (default); the *Wrap Text* box displays; check this box if you want text to wrap and stay within the cell.
  - **Date**; In the *Format* field that displays, type the format you want to use (Excel formatting is supported):
    - 24-hour format: m/d/yy h:mm:ss
    - AM/PM format: m/d/yy h:mm:ss AM/PM or m/d/yy hh:mm:ss AM/PM
  - **Number**; In the *Decimal Places* field that displays, choose the number of decimal places you want; if desired, check the *Use 1000 Separator* box to insert the separator (for example, comma, depending on your regional settings).
- You can resize the row height or column width by dragging row/column header. A tooltip displays the height or width as you resize it.

Alternatively, right click anywhere in the template to display a context menu that allows you to insert/remove columns or rows, or to type the column width and row height.

Save your changes.

**See also:**

To add data formulas to the Live View template, see [LiveView Formulas on page 213](#).

To add data (device names and tag names) to the Live View template, see [LiveView Placeholders on page 212](#).

To add visual alerts (color changes) when the value of the tag associated with a cell becomes too high or too low, see [LiveView Thresholds on page 214](#)

See also:

[LiveView Formulas on page 213](#)

[LiveView Thresholds on page 214](#)

[LiveView Placeholders on page 212](#)

## LiveView Placeholders

Placeholders provide the data (device names and tag names) to a LiveView template. The placeholders are the identifiers that are added when setting up the template, but are replaced with the name of the selected device or the tag value when the template is viewed.

To use this feature, click the Placeholder subtab in LiveView Template Editor. Then enter the appropriate information in the following fields.

1. To begin, place the cursor in a cell. Note that the Insert Location displays the cell number for the placeholder you are setting.
2. **TAG Value/Device Name:**

From the drop-down field in the top left corner of the page, choose either Tag Value or Device Name.



**Tag Value:** Select the tag group, such as Alarm, Current, Energy. Beneath the tag group, select the specific tag you want. The list is filtered to include only the most common tags that belong to the group you selected. To view all of the tags available in this tag group, check **Show Advanced**.

**Device Name:** The list of devices is filtered to include only devices for which this template's data is available. To display the device name in this cell of the template, select Device Name. You will choose the actual device during runtime.

3. **Insert Location:** This offers a second way of inserting the placeholder location. After choosing the device or tag, type the cell number for the placeholder cell.
4. **Insert:** Click to add the selected placeholder to the specified cell.

Continue adding placeholders as needed.

See also:

[LiveView Formulas on page 213](#)

[LiveView Thresholds on page 214](#)

[LiveView Formatting on page 211](#)

## LiveView Formulas

Follow these instructions to add the formulas needed to include data in a Live View template. You can add formulas to:

- add, subtract, multiply, or divide the contents of two individual cells
- add, multiply, or average the contents of a range of cells

To use this feature, click the Formulas subtab in LiveView Template Editor. Then choose one of the following fields:

- **Cell:** Use this field to enable a formula for two individual cells. Then enter:
  - Cell 1 Address:** Type the cell address. The cell address displays in this field.
  - Operator:** Choose the operator you want to use: +, -, \*, or /.
  - Cell 2 Address:** Type the cell address. The cell address displays in this field.
- **Cell Range:** Use this field to enable a formula for a range of cells. Then enter:
  - Operation:** Choose average, product, or sum.
  - Cell Range:** Type the cell range (format C4:C20), or select the range of cells to include in the formula. The cell range displays in this field.
- **Insert Location:** Type the cell number.
- **Insert:** Click this button to build the formula you have specified, and to add it to the cell you added to Insert Location.

Repeat the above procedure for the rest of the formulas you want to use for this Live View template.

### NOTES:

- You must "Protect Current Sheet" in order for formulas to be maintained and visible in the LiveView Template Editor.
- If you want to use conditional formulas ("IF" formulas), you must first create them in Excel. To do this, you must access the template you want on the server (Program Files > Schneider Electric > Applications > LiveView > TemplateEditor > Templates Temp). Open the template in Excel and add the conditional formula(s) that you want. After you save the changes, the formula(s) will function correctly in Live View. You must copy the IF statement into every cell of the column that displays the result of the IF statement.

- In multiple device tables that rely on formulas to display information for each device, the results column will display zeroes when that row has no device in it. To avoid this, use a formula that will display no result if there is no device in that row. In the following example, when no device is in cell A2, no results will display (no zeroes) in cell E2.

	A	B	C	D	E
1		Value 1	Value 2	Value 3	Sum
2	<<"Dn">>	<<"POWER:1039">>	<<"POWER:1040">>	<<"POWER:1041">>	=IF(ISBLANK(A2),"",SUM(B2:D2))

**See also:**

[LiveView Placeholders on page 212](#)

[LiveView Thresholds on page 214](#)

[LiveView Formatting on page 211](#)

## LiveView Thresholds

Use this feature to display tag readings that fall outside of the normal range. You can apply it to an individual cell or a range of cells. You determine the tag or tags for which you want to display out-of-normal (threshold) readings. When the value of the tag in a cell (or any tag in a cell range) is below the minimum or above the maximum that you set, the tag value displays in the threshold cell.

You can set both minimum and maximum values for a cell or cell range. Use different colors to indicate the high and low readings.

To use this feature, click the Threshold subtab in LiveView Setup. To add a threshold:

- Cell:** For a single cell: Select the cell for which you want the font color to change. The font color will change when the value for the tag in that cell goes above the specified Max Value (or below the Min Value) for the threshold.  
or  
**Cell Range:** For a range of cells, either select the range, or type the range in the format C4:C20.
- NOTE:** When setting up a multiple-device table, you should use a cell range to ensure that threshold font colors display for each device in the table.
- Min Value:** Type the low value for the "normal" range. If the tag value drops below this value, the cell font color will change as specified in step 3.
- Below Min Threshold Color:** Open the color palette and select the font color that you want to indicate the "low" status.
- Max Value:** Type the high value for the "normal" range. If the tag value goes above this value, the cell font color will change as specified in step 5.
- Above Max Threshold Color:** Open the color palette and select the font color that you want to indicate the "high" status.
- Insert Location:** Choose an empty cell, one that is not part of the table. This cell will be the location for the threshold definition that you are creating.

The default cell for the threshold definition is the next available cell in the template. For example, if the tag in cell B7 has an unused cell to the right of it (C7), the threshold definition defaults to C7. Then, when the value in B7 exceeds the threshold defined in C7, the value in B7 displays in the font color you specified. To override the default cell location, change it in the Insert Location field.

7. **Insert:** Click here to create the threshold(s).

The threshold definition is in the form: <<Threshold;B2:B20;Min=100;Max=1000>>


See also:

[LiveView Formulas on page 213](#)

[LiveView Placeholders on page 212](#)

[LiveView Formatting on page 211](#)

## Modify LiveView Template

You can modify any template except one that is predefined. Predefined templates have a lock icon (  ) beside their names.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 210](#)). In the PLS\_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to modify.  
The template displays.
3. You can change any field on the template. Click any of the subtabs (Placeholder, Formula, Threshold, or Formatting) to edit the related information. For help on the subtabs, see the "See Also" links below.
4. When you have finished making changes, click Save.

Continue working with other templates.

See also:

[LiveView Placeholders on page 212](#)

[LiveView Formulas on page 213](#)

[LiveView Thresholds on page 214](#)

[LiveView Formatting on page 211](#)

## Duplicate LiveView Template

Use this feature to make an exact copy of any template, including predefined templates. The duplicate will not be locked, allowing you to edit and save it as a different template.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 210](#)). In the PLS\_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to duplicate.  
The template displays.
3. Click Duplicate (on the top of the left-hand pane).  
The duplicate template is added to the list. It has the same name of its original template, appended with "Copy."
4. Change the name of the duplicated template to differentiate it from its original.
5. Make the desired changes and then click Save to save them.

## LiveView Delete

You can delete any template except one that is predefined.

1. Open LiveView Template Editor. In Runtime mode, click the menu links that have been set up when you created the graphics page (see [Create Menu Item for LiveView Page on page 210](#)). In the PLS\_Example project, there is a tab for LiveView.
2. Highlight the name of the template that you want to delete.  
The template displays.
3. Click Delete (on the top of the left-hand pane).  
You are prompted to verify the deletion.
4. Click Yes to delete the template, or click No to cancel the deletion.

Continue working with other templates.

## LiveView Template Localization

You can easily localize your LiveView templates. To do this, you will need to replace English terms with the localized ones. Follow these instructions to create the localized file:

1. Choose the language that you need for your localized terms.
  - Go to Control Panel > Internet Options > Languages. In the Language Preference box, click Add to display the possible languages.
  - Highlight the language you need. For example, French for France is: **French (France) [fr-FR]**. Click OK.
  - In the Language Preference box, highlight the new language and click Move up to move it to the top of the list of languages. Click OK.
2. Locate the file called LiveViewViewer.resx. The default location is in the App\_GlobalResources folder.
3. Make a copy of the resx file.
4. Use Visual Studio to open the copy resx file.
5. Replace the English terms in the right hand column with the localized terms that you need.
6. Save this file with a new extension, which must match the language. For the example above, the new file name would be:

**LiveViewViewer.fr-FR.resx**

The LiveView template fields that you replaced will now display in the localized language.

## Enable Windows Authentication for LiveView

You may wish to use Windows Authentication for logging in to LiveView. If you want to use Windows Authentication, you must follow standard IIS authentication methods. Follow the steps below to configure Windows Authentication.

**NOTE:** These steps are specific to Windows 7; they may be different for other operating systems. For further assistance, view Microsoft's documentation on this topic at:

[http://technet.microsoft.com/en-us/library/cc754628\(v=ws.10\).aspx](http://technet.microsoft.com/en-us/library/cc754628(v=ws.10).aspx)

1. Turn on the Windows Authentication feature:

- a. From the control panel, click Programs and Features > Turn Windows features on or off.
- b. Check Windows Authentication.
2. Enable Windows Authentication in IIS:
  - a. From the Control Panel > Administrative Tools, choose Internet Information Services (IIS).
  - b. Select the root node from the tree on the left (or the LiveViewViewer node, if this server hosts multiple sites).
  - c. From the right hand pane, in the IIS section, click Authentication.
  - d. Enable Windows Authentication.
3. Modify web.config to specify Windows Authentication:
  - a. Locate web.config:  
C:\Program Files\Schneider Electric\PowerSCADA Expert\v8.2\Applications\LiveView\Viewer
  - b. Change the line:  
<authentication mode="Forms">  
to  
<authentication mode="Windows">
4. Add roles to web.config:

You must add roles to web.config to allow access to the LiveView application. For example, to allow the role (group) Administrators, add the following to the web.config file:

```
<authorization>  
  <allow roles="Administrators"/>  
  <deny users="?" />  
</authorization>
```

Modifying the web.config file is an advanced topic that is covered on the Microsoft website.

You will enable Windows Authentication in IIS and modify web.config to specify Live View.

See the following link for instructions provided on the Microsoft website:

<http://www.iis.net/configreference/system.webserver/security/authentication/windowsauthentication>

Additional information may be available in the following knowledge base article:

<http://support.microsoft.com>, and search on kb/815179.

## Rapid Access Labels

Use this report to create quick-response code (QR code) stickers that can be placed on your system equipment to provide quick access to PowerSCADA Expert standard reports and LiveView table views. You can also generate a label for any URL. After you create and print the code stickers, you will be able to read them with a QR code reader that is installed on a Smart Phone, iPhone, or other device.

## Before You Begin

Make sure that you have completely configured your system. This includes:

- Set up all servers, equipment, and addressing

- Create the reports and LiveView views that you want to see. Note that the report configuration names and Live View view names cannot be changed after you create the QR stickers, as the links would be broken to the reports/table views.
- To view a report, you must first save the report configuration. See Run a Report and Save its Configuration in [Create and View Basic Reports on page 189](#) for instructions.
- All devices that will be used to scan QR codes must be on the same network with the server.
- Purchase the label stock paper for the labels you will print. Use **Avery 6578 Label Stock**, or equivalent. Other label stock may not be spaced correctly, which could result in the labels not printing correctly.

## Create the Sticker

1. Ensure that you have a laser printer set up and available for printing.
2. At the printer that you will use to print the labels, insert the blank label stickers. Use **Avery 6578 Label Stock** or equivalent.
3. Browse to the Rapid Access Labels report in the reporting web application. When prompted, enter your PowerSCADA Studio user account information. Click Login. The Rapid Access Labels screen displays.
4. From the *Server Address* drop down list, choose the IP address that is connected to the same network as the wireless access points. Do not use an IP provided by DHCP, as the IP address can change frequently. If your network supports DNS, we recommend that you use the machine name of the server.
5. In the *Port* box, accept the default "80" or, if necessary, enter a different port.
6. In the *Select items to generate labels* box, check the report configuration(s) and LiveView table(s) for which you want to print stickers.
7. Optional: You may wish to print a sticker for a different URL (such as a corporate website). To do this, enter the URL in the *URL* line of the *Manual URL Entries* box (the site name automatically displays in the upper box).
8. Optional: On the *Caption* line, you can type any text that you want to have printed above the QR code on the sticker. If you want the output table or report to have a title, enter it here.
9. Click *Generate Report*.

NOTE: To print correctly, use the icon on the report control bar, not the one from the browser (which would add a header and footer, and throw off alignment).

## Read the Sticker

The sticker(s) print at the designated printer. Each one has a title that is one of the following:

- a report configuration name
- a LiveView name
- user-entered text from the *Caption* text box

Place each sticker in the desired location, such as next to the device that is being monitored.

To read a sticker, use a QR code reader (such as a Smart phone or iPhone). The reader must have access to the network and server. We recommend that you use the QR Droid application if you are viewing reports/tables from an Android phone.

## Troubleshooting

If you cannot read the QR code, verify the following:

- Your phone/reader has access to the wireless network, and the server can be reached by the IP you selected when generating labels.
- The server address and port name are correct.
- The report configuration name or LiveView table name are correct (and have not been changed or deleted).

## Multi-Monitor Support

PowerSCADA Expert provides the means for viewing the runtime environment from multiple monitors. This setup allows you to view unrelated information at separate monitors that are placed beside each other. In this example, there are four monitors, configured left to right in a single row:

### Main Monitor Monitor 2 Monitor 3 Monitor 4

(displays Windows Start button)

In the Citect.ini file for the project, add the following instructions:

Add the following instructions to the Citect.ini file (changes can be added directly to the Citect.ini file):

```
[MultiMonitors]
```

```
Monitors = 4
```

```
StartupPage1 =
```

```
StartupPage2 =
```

```
StartupPage3 =
```

```
StartupPage4 =
```

where

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

The windows display according to the formation of the monitors—such as left-to-right, top-to-bottom or grid—set in the Windows operating system. By default, this is set via the Windows Start menu > Control Panel > Display applet.

StartupPage1 = startup page that needs to be displayed on monitor 1

StartupPage2 = startup page that needs to be displayed on monitor 2

StartupPage3 = startup page that needs to be displayed on monitor 3

StartupPage4 = startup page that needs to be displayed on monitor 4

## The Demo Mode

You can run PowerSCADA Expert without the hardware key in demonstration (Demo) mode. In demonstration mode, you can use all product features normally, but with restricted runtime and I/O.

If you configure PowerSCADA Expert to run as multiple processes on one CPU or multiple CPUs, you cannot use it in demo mode; but if you run it as one process, you can use a demo mode. The following demonstration modes are available:

- The one-line engine will run for 60 minutes.
- Reporting and LiveView do NOT work, because it requires a valid license.
- 15 minutes with a maximum of 50,000 real I/O.

- 10 hours with a maximum of one dynamic real I/O. This is useful for demonstrations using memory and disk I/O. PowerSCADA Expert starts in this mode if no hardware key is available. If the system detects that you are using more than one real I/O point at runtime, it will switch to the 15-minute demo mode.

Writing to any tag through DDE, CTAPI, or ODBC will cause that tag to contribute to the dynamic point count, even if it is a memory or disk I/O point. So, if you write to more than one point through these interfaces, it will switch to the 15-minute demo mode.

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

## Application Configuration Utility

Use the Application Configuration Utility to easily configure many features that would require more time-consuming effort if performed via INI settings. See [Add INI Settings to AdvOneLine.ini and Citect.ini on page 137](#) for details about these settings.

Options that are available on every page are:

**Project Name:** Located at the top of the page, this option allows you to choose the project. Unless you change it, this project will then remain selected for each window in the Application Configuration Utility.


**Restart Services:** This feature is available only when changes are made to the Citect Data Platform page. Click this link to restart Application Services. If you click inside any field of this area and then click Display Selected or Display All, you are prompted to save your changes before the configuration file displays.

**Expander Toggle:** Click the triangle that is on the vertical bar that separates the left and right panels of each screen (shown below). This toggles open or closed the menu in the left-hand column.



**Display Selected:** Click this link to display the settings that have been entered in specific area of the Application Configuration Utility (Application Services, Application Services Host, Applications, Security) that you are viewing.

**Display All:** Click this link to view the settings that have been entered for the entire Application Configuration Utility.

**Search:** Click this link to open a search window. Type the key word or phrase you want to search on, then click  to view the list of screens on which the word or phrase are found. Click a screen name, and the screen displays. Click the 'x' in the upper right corner of the search results to close the search window.

**Tooltips:** To view help for an individual field, point your mouse and hover over the field.

The Application Configuration Utility includes these sections:

**Application Services:** The [Application Services on page 221](#) section includes screens for setting up:

- [Diagnostics on page 221](#)
- [EcoStruxure Web Services Setup on page 222](#) (EWS)
- [Event Notification Service on page 222](#)



**Application Services Host:** This includes:

[Application Services Host—Citect Data Platform on page 223](#): four tabs contain settings to configure server/user name, ignored devices and topics, deadbands, and to display the license of the server.

**Applications:** This includes:

[Basic Reports on page 225](#): Use this screen to set up the delivery mode and email address from which PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards basic reports will be sent.

[One-Line Engine on page 224](#): Two tabs contain settings to configure one line engine behaviors.

**Diagnostics page:** The [Diagnostics Page/I/O Device Settings on page 225](#) provides a quick view of the I/O device INI settings for all protocols, clusters, servers, ports, and devices. Use this information as the first step in troubleshooting device/communication issues in your system.

### Security

- [One-Time Password \(Two-Factor Authentication\) on page 226](#)
- [Single Sign On on page 235](#)

## Application Services

Use this section to configure services that connect with PowerSCADA Studio. There are three tabs:

[Diagnostics on page 221](#): This is a read-only list of web services hosted by the Schneider Electric CoreServiceHost, details about them, and whether they are running. You can also choose the logging level.

[EcoStruxure Web Services Setup on page 222](#): EWS uses web-based http protocol to transfer data. It enables two-way data transfers, which allows the acknowledgment of alarms from SBO.

[Event Notification Service on page 222](#): This feature serves alarms and events to the Event Notification Module application through a web service endpoint named Alarms/Alarm Proxy.

### Diagnostics

#### Logging Level:

This feature turns on extra diagnostic information that can be useful when diagnosing problems that occur in application services or its hosted applications (such as LiveView). Choose the level of logging to be used in all applications. Debug and Verbose increase the amount of information that is logged during runtime for applications such as Basic Reports and LiveView.

- Normal: Use when the project is live.
- Debug: includes additional logging statements (in the Windows event log named PowerLogic). This logging should not affect performance in the system during runtime.
- Verbose: releases additional diagnostic information, such as large lists, that could affect system performance.

#### Service Inventory:

This is a read-only list of web services hosted by the Schneider Electric CoreServiceHost, details about them, and whether they are running.

## EcoStruxure Web Services Setup

This feature configures the PowerSCADA Expert EcoStruxure Web Services (EWS) server. (See [EcoStruxure WebServices \(EWS\) on page 186](#) on page 1 for a description of this server.) Do not confuse this information with the EWS server that was released with PowerSCADA Expert 7.40. That implementation is specific to the PowerSCADA Studio core. It was developed only for real-time tag data acquisition. The implementation being released with this product also acquires historical data and alarms.

The fields are:

- **Alarm Acknowledgment Wait Period:** The amount of time allowed for PowerSCADA Expert to process an alarm acknowledgment request. Choose a value that allows the system enough time to allow acknowledgments to be processed, while not so long as to delay processing.
- **Initial Alarm Request Length:** The number of days worth of alarm to request from PowerSCADA Expert.
- **Max Request Size:** The number of alarms returned with one request. The default (1000 alarms) should be sufficient to maintain alarm data integrity (ensuring that all alarms are returned in each call), while also maintaining system performance.
- **Alarm Settle Time:** The number of seconds "grace period" to allow the PowerSCADA Studio Alarm Server to finish inserting alarms that are in process at the time of the poll. If you set this too low, you could miss alarms. If you set it too high, it may take longer for alarms to come into EWS.
- **EWS/Citect User Association:** Use this block to manage user names and passwords. This provides EWS Digest Authentication for the user, permitting them to view data. However, for the user to be able to acknowledge alarms, the username/password must match a username/password added to the PowerSCADA Expert project. When this user acknowledges an alarm through EWS, PowerSCADA Studio verifies the credentials of the user and acknowledges the alarm under this user's identity.

To add a user, click Add. At the Add User screen, type an established PowerSCADA Studio username and password. Click *Test Citect Credentials* to verify the name and password. When you enter a valid username and password, a message displays telling you they are valid.

## Avoiding EWS Provider Timeouts

When you attempt to retrieve a large amount of data from the EWS server, you could see an error telling you that the call has timed out. To correct this, you can temporarily increase the timeout period, which will allow the target application to receive the data. To do this, modify the key named *ProviderTimeoutInMinutes*, found in the EWS virtual directory, under Web.config.

Default location;

```
C:\Program Files (x86)\Schneider Electric\PowerSCADA  
Expert\v8.2\Applications\EWS\Web.config
```

After the data that you want is processed, you need to return this key to its original setting.

## Event Notification Service

This feature serves alarms and events to the Event Notification Module (ENM) application through a web service endpoint named Alarms/Alarm Proxy. The fields are:

- **Maximum Number of Alarms/Events per Call:** This field controls the maximum number of alarms or events that can be requested in one poll. As the requesting client continues to request data, additional data will be served at the same rate.
- **Poll Period:** The frequency at which the Event Notification Service polls runtime for alarms (in seconds).
- **Alarm Settle Time:** The number of seconds "grace period" to allow the Alarm Server to finish inserting alarms that are in process at the time of the poll. If you set this too low, you could miss alarms. If you set it too high, it may take longer for alarms to come into ENM.

## Application Services Host—Citect Data Platform

This section relates to how the Schneider Electric CoreServiceHost connects to PowerSCADA Studio.

Use this page to link a PowerSCADA Studio user name and password to be used when the Schneider Electric CoreServiceHost services connect with runtime. Before you begin:

- Add the username/password to the PowerSCADA Studio project.
- Have the project running in runtime mode.

Follow these steps:

1. In the Citect .ini file, set [ctAPI] Remote = 1
2. **Citect I/O Server Address:** Choose the server address for the project that is running.
3. **Citect User Name:** Type the user name for this user.
4. **Citect Password:** Type the password for this user.
5. **Test Credentials:** Click to verify these credentials. If you see an error, verify the name and password, and that runtime is running, and try again.

When your project is running and the credentials are valid, you see Connection Successful. The user name and password can be used to connect to PowerSCADA Studio.

**Citect Licensing Details:** This is a read-only field that displays the license key currently in use on the server machine.

### NOTE:

To provide extra security, you can use the following method. You will run as a Service. PowerSCADA Studio and services must both run on Session 0.

1. In the Citect .ini file, remove [ctAPI] Remote or set it to 0.
2. Leave **Citect I/O Server Address** blank.
3. Leave **Citect User Name** blank.
4. Leave **Citect Password** blank.
5. When you **Test Credentials**, the test will fail. However, you can verify that the service has started by viewing the Event Log.

## Applications

Use this section to set up information for:

[Basic Reports on page 225](#)

[One-Line Engine on page 224](#)

## One-Line Engine

To access this feature, click Start > All Programs > Schneider Electric > PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards > Config Tools > Configurator. There are two tabs in the One-Line Engine module. On either tab, two buttons at the bottom allow you to:

**Restart AOL:** Click to manually restart the Advanced One Line engine.

**Save:** Click to save the settings you have entered.

**General:** You can edit the following parameters that enable one lines to run properly. For more complete descriptions of the parameters, see [Add INI Settings to AdvOneLine.ini and Citect.ini on page 137](#). If you are not setting up a redundant system, the default settings should be sufficient.

- **Update Interval:** interval in seconds at which the system tries to solve the system one-line
- **Max Startup Delay:** sets the amount of time in seconds the AdvOneLine.exe has to start up
- **Health Timeout:** performance parameter; dictates the amount of time in seconds that must elapse before the one-line engine is considered to be non-functioning
- **Log File Length:** suggests the log file length in number of lines
- **Debug Level:** selects the level of logging for AdvOneLine.exe

### Redundancy:

- **Primary Server IP:** used in redundant configurations to specify the IP address of the primary IO Server. Click Clear to remove the current address, then type the correct address for the primary server.
- **Standby Server IP:** used in redundant configurations to specify the IP address of the standby IO Server. Click Clear to remove the current address, then type the correct address for the secondary server.

Use the following two buttons to save an encryption key and export it to another computer as an .aes file.

## NOTICE

### LOSS OF SYSTEM CONFIDENTIALITY/INTEGRITY/AVAILABILITY

Do not save the .aes encryption file to the local computer. Always export it to a secure site, such as a secure network or to the target computer.

**Failure to follow these instructions can result in unauthorized access to, modification of, and unavailability of the system.**

This allows you to move an INI file from one computer to another and to have its contents unencrypted for use by that computer.

- **Export Key:** After making/verifying changes here or in the AdvOneLine.ini file, click to save a configuration that you can use on another computer.

A Save As window displays, allowing you to browse to the preferred location. Save the .aes to a secure location, such as a secure network drive or a flash drive.

- **Import Key:** After you save the .aes file to the secure drive, ensure that the drive is accessible to new computer. At the new computer, click this button to access the .aes file.

After you access the .aes file at the new computer, copy the INI file to the new computer. You will be able to access and use it.

Remove the .aes files from the source computer.

### Basic Reports

Use this screen to set up the delivery method and email address from which PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards basic reports will be sent. These settings specify the SMTP server for emailing basic reports. They are not used for Event Notification Service.

Define the following:

- **Timeout:** The number of seconds PowerSCADA Expert will attempt to deliver an email before no longer attempting
- **Delivery Method:** Network (default), Pickup Delivery from IIS, or Specified Pickup Directory. This is an SMTP-specific setting. In most cases, use Network. For more information on SMTP, see the Micrologic Developer Network website.
- **'From' Address:** the address from which reports will be sent.
- **Host:** The IP or network address of the SMTP server.
- **Port:** The network port to be used; default for SMTP is 25.
- **Use Default Credentials:** If required by the SMTP server being used, uncheck the box and enter the appropriate user name and password. If not required, check the box and enter the SMTP user name and password used for reporting.

## Diagnostics Page/I/O Device Settings

The Diagnostics page provides a quick view of the I/O device INI settings for all protocols, clusters, servers, ports, and devices. Use this information as the first step in troubleshooting device/communication issues in your system.

When you click the Diagnostics tab, a short description and help link display on the right-hand side. The basic steps you follow are below. See the section after the steps for the logic behind how data displays.

1. To view data, click the Diagnostics link in the left-hand pane of the Application Configuration Utility. Then click the I/O Device Settings sub-link.
2. Ensure that your project is selected from the Project Name link at the top of the page.
3. From the Servers column, click a cluster and then a server.  
Note the server type on the bottom line of each server. You could have multiple servers: I/O, alarm, and trend.
4. (I/O servers only): A "Component" column displays. From this column, choose a protocol and then the port and device.
5. View the data for that server/device:
  - The top row of the Settings are a "breadcrumb," showing the device information from cluster to device.
  - The second row, just above Effective Settings, displays details about the device, equipment name, number of tags, address, priority, memory mode and startup mode.
  - The remaining sections display information only if there have been changes made to the default settings.

### Data Selection Logic

Depending on the server type that you select, the Settings on the right display with different information.

All three server types display settings that include:

- **Default Settings:** the device default before any changes have been made; displays above the Effective Settings section.
- **Effective Settings:** the current settings, including any changes made, combining changes from the protocol, port, and device. In all cases, port changes will override protocol changes, and device changes will override port changes.
- **Protocol Settings/Port Settings/Device Settings:** If there are any overrides to the default settings, they display here.

I/O servers alone display a separate column: When you select an I/O server, the Protocol column displays beside it. You will select protocol, port, and device.

Data is retrieved in this manner:

- Protocol settings are retrieved from the Param.dbf file and then from the citect.ini file.
- These two lists of settings are merged. If there are duplicate settings, the citect.ini file changes take precedence.
- Finally, device settings are retrieved and merged. If there are duplicate settings, the device settings take precedence.

For example, for a given device:

The Protocol Setting for CacheRefreshTime = 2000.

The Port Setting = 1000.

There is no additional Device Setting to override the other settings.

The Effective Setting uses the CacheRefreshTime of 1000 from the Port Setting.

## Security

Use this section to set up information for:

*One-Time Password (Two-Factor Authentication)*

*Single Sign On*

### **One-Time Password (Two-Factor Authentication)**

Two-factor authorization (2FA) requires users to provide two pieces of proof of identity, such as a password and one other component. This feature allows you to add an additional layer of protection when user credentials are required (for example, at login and shutdown).

Traditionally, two-factor authentication requires a password and another piece of identity, such as an email address or a phone number (with SMS texts). For example, some websites require the user to provide a password followed by a code that is sent via text message or email.

PowerSCADA Expert uses a one-time password (OTP) to accomplish two-factor authentication. The OTP mechanism eliminates the need for email or text messaging. The OTP/2FA can be deployed on a completely isolated network.

OTP is implemented in PowerSCADA Expert using a small USB-key device called a YubiKey. The YubiKey device is designed to fit on a key ring or attached to a badge. It must be plugged into the client machine when the user authenticates. There are multiple models of YubiKeys available. Included is one model with Near Field Communication (NFC) capability in the event that USB ports cannot be used.

To log in, the user enters their user name and password. PowerSCADA Expert then prompts the user to press a button on the YubiKey device (which has been plugged into a USB port on the client). The YubiKey will send a string of characters (the OTP) to PowerSCADA Expert for authentication. If both the password and the OTP authenticate successfully, then the user is logged in. Note that the OTP is only authenticated successfully one time. Once successful, the OTP expires. This means that if the string is captured (snooped from the network, for example) and reuse is attempted, the OTP will fail.

**How Does It Work?**

A YubiKey is assigned to a PowerSCADA Expert user (see below). After that, the YubiKey and the assigned user account share a secret code. The YubiKey uses this secret code to generate encrypted strings of text (the OTPs) when the user presses the button on the YubiKey.

Using the secret code, PowerSCADA Expert decrypts the OTP to determine if the OTP is valid (ensuring that it has not been replayed, it is assigned to the current user, etc.). After successful authentication, PowerSCADA Expert marks the OTP as expired and will no longer accept it as valid.

**Select the YubiKey**

YubiKeys are not shipped with PowerSCADA Expert. Instead, you must buy them from a third-party vendor, such as Amazon. The following table describes the various YubiKey models that are compatible with PowerSCADA Expert.

Model Number	Description
YubiKey Standard	Standard YubiKey, suitable for most deployments
YubiKey Nano	Small form factor, designed to remain inserted in the USB port. Suitable for use in clients with a dedicated single user account.
YubiKey Edge	Similar to Standard, includes advanced features not useful for PowerSCADA Expert. Recommend YubiKey Standard instead. May not support auto configuration.
YubiKey Edge-N	Similar to Nano, includes advanced features not useful for PowerSCADA Expert. Recommend YubiKey Nano instead. May not support auto configuration.
YubiKey NEO	Similar to Standard, includes NFC capability. Note: NFC setup requires Yubico personalization tool (free) and a compatible NFC reader.
YubiCo NEO-N	Similar to Nano, includes NFC capability. Note: NFC setup requires Yubico personalization tool (free) and a compatible NFC reader.

For more information about these keys, see <http://www.yubico.com>.

**Deployment Guidelines**

Keep in mind these points when you are ordering or using a YubiKey.

- You must set "Allow RPC" to TRUE for all roles that are using YubiKey.
- YubiKey is compatible with all thick clients and web clients.
- YubiKey requires access to a USB port at each client (or must include NFC capability).
- While technically compliant, NFC has not been validated for use with PowerSCADA Expert.
- Each PowerSCADA Expert I/O server must have Application Services (Core Service Host) running.

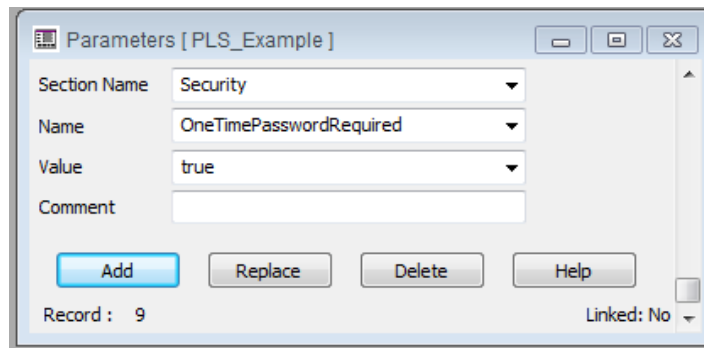
- Multiple I/O servers may reside on a physical machine. In this case, only one instance of Application Services resides on the machine.
- Yubikey must be configured and synchronized across all I/O Servers (this includes redundant pairs and distributed systems).
- YubiKey is enabled on each client independently.
- It is possible to configure YubiKey on one machine, export the configuration for all users, and import the configuration to all remaining machines.
- It is not necessary to re-program YubiKey when changing passwords. The YubiKey constantly changes the OTP (as it is only valid once); so it is not susceptible to replay attacks.
- YubiKey is authenticated against all physical servers that contain at least one I/O server. All physical servers must successfully authenticate the OTP for success. If a single physical server does not authenticate (due to misconfiguration, etc), the user will not be able to log in.
- If a physical machine (with an I/O Server) is not available, it is not included in the authentication scheme (this means that, if a primary server is down, the secondary can still successfully authenticate the OTP).
- If no physical servers (with I/O Servers) are available, the user will not be able to log in on clients that have YubiKey enabled.

## Add the Citect Parameter

You need to add the parameter that allows PowerSCADA Expert to communicate with the YubiKey. You can do this before after you configure the YubiKey.

From the PowerSCADA Studio, click Setup; from the top toolbar, choose Parameters. At the Parameters screen, type the following:

- Section Name: Security
- Name: OneTimePasswordRequired
- Value: true



Click Add, then compile the project to add the parameter.

To be able to use YubiKey in PowerSCADA Expert, you must set "Allow RPC" to TRUE for all roles that include users with assigned YubiKeys. The default for PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards is False. To change this to True, access the user account through Roles in the PowerSCADA Studio, Security tab. Change "Allow RPC" to **TRUE**.

## Autoconfigure the YubiKey

To avoid the lengthier process of programming the key, in most cases you can choose to autoconfigure it.

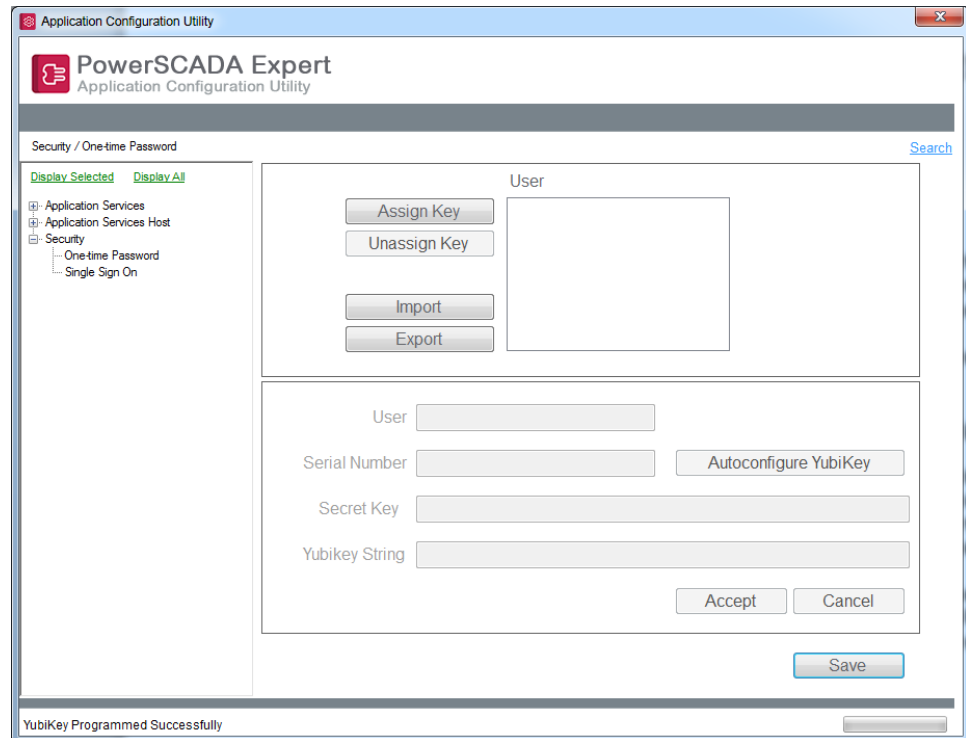


**NOTES:**

- This procedure requires that you have a USB port on your computer. It will not work for virtual machines.
- An option for computers without USB ports is to autoconfigure on a local workstation, then export the settings to a shared drive and import the settings onto the computer where you need them. (See [Import and Export One-Time Password Settings on page 233](#), below.)
- You can only have one YubiKey inserted at a time.
- If Autoconfigure will not work, and you must manually program the YubiKey, see [Program the YubiKey on page 230](#) for instructions.

To autoconfigure, follow these steps:

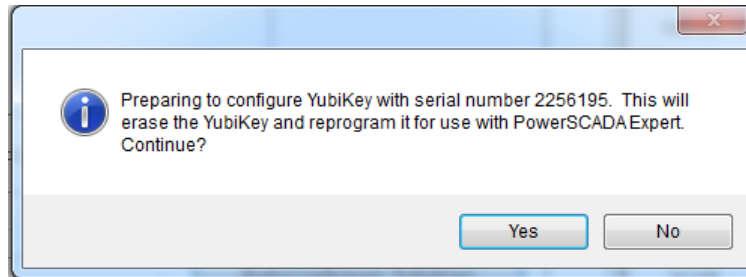
1. Insert the YubiKey into the USB port of the computer.
2. In the Application Configuration Utility, navigate to the One-Time Password screen from the Security menu:



3. Click Assign Key.

The grayed-out fields become live.

4. In the User field, type the PowerSCADA Expertusername to which you want to assign the YubiKey.
5. Click Autoconfigure YubiKey (in the lower half of the screen). You see this message:



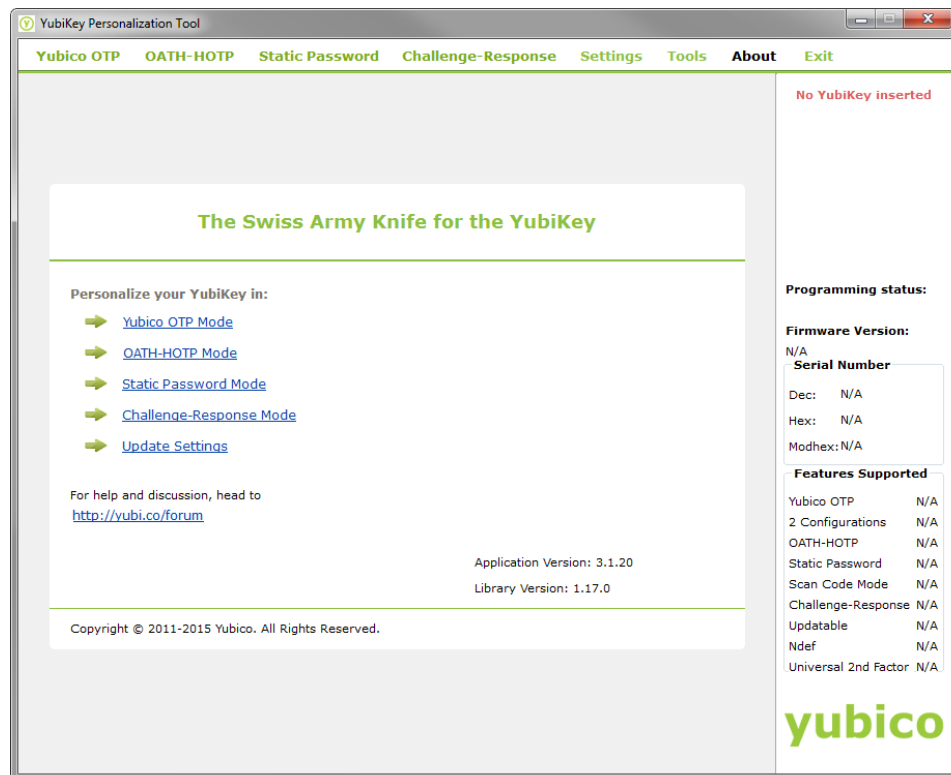
This message tells you that all settings on the key will be erased, including any key assignments.

6. To continue, click Yes. The key will receive a new secret key.
7. Click Accept.

## Program the YubiKey

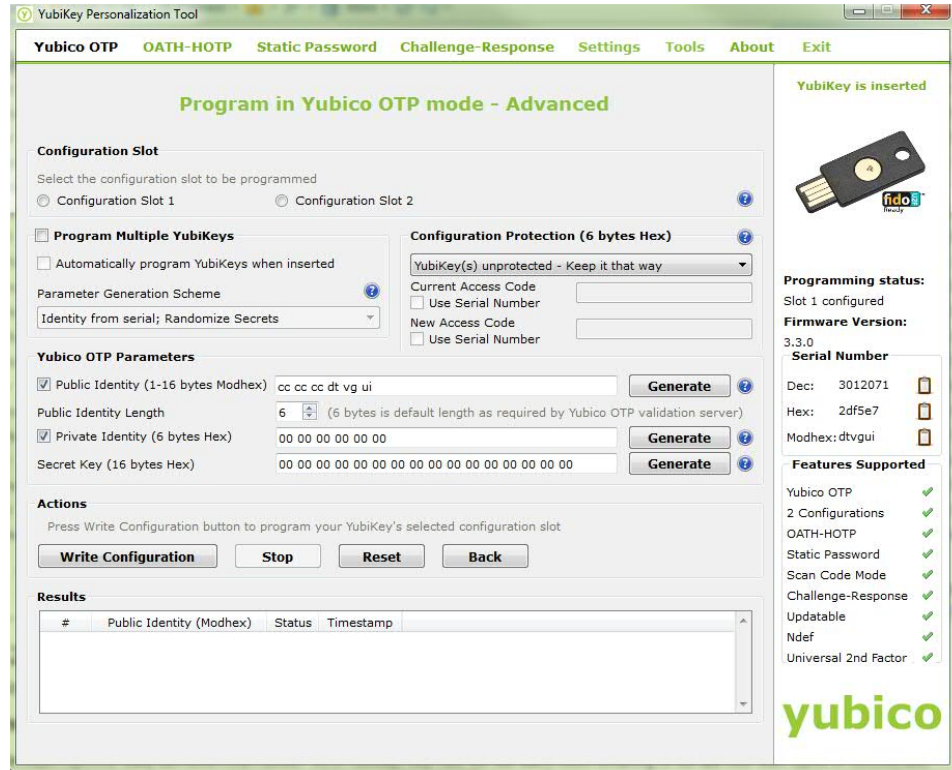
If you cannot autoconfigure, use this procedure. After you obtain the YubiKey from a third-party vendor, (such as Amazon), you need to download the YubiKey Personalization Tool from the Yubico website ([www.yubico.com](http://www.yubico.com); click Products > Services & Software > Personalization Tools > Download Yubikey Configuration Tools).

1. Download and install the YubiKey Personalization Tool.
2. Launch the tool, and this screen displays:



3. Insert the YubiKey into a USB port of your computer. Click the *Yubico OTP Mode* link. At the next screen, click *Advanced*.

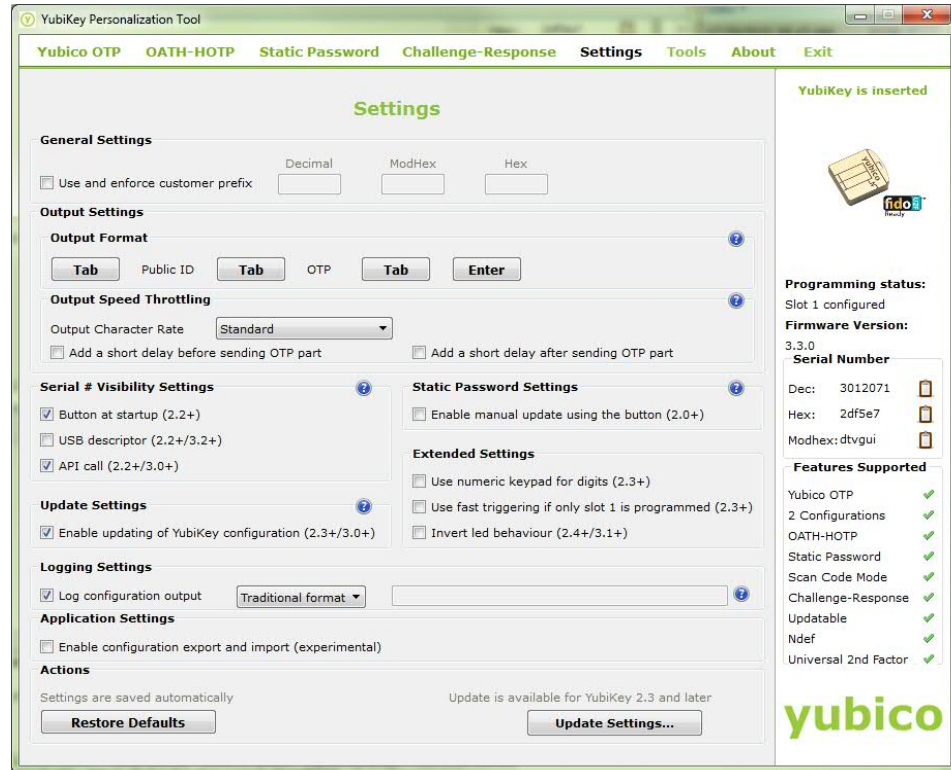
This screen displays:



If you want to use both of the key's configuration slots, download the YubiKey documentation, found under the Support tab of the Yubico website.

To configure a single slot, follow these steps:

4. Click the checkbox for the slot you want to configure.
5. Under Yubico OTP Parameters, check Public Identity. Click Generate.
6. Leave the Public Identity Length at its default, 6.
7. Check Private Identity. Click Generate.
8. Click Generate beside Secret Key.
9. Make a note of the secret key that displays, including all characters and spaces. You will need it when you add the key to the Application Configuration Tool.
10. Click Write Configuration.
11. Click the Settings tab. This screen appears:



Enter the following information:

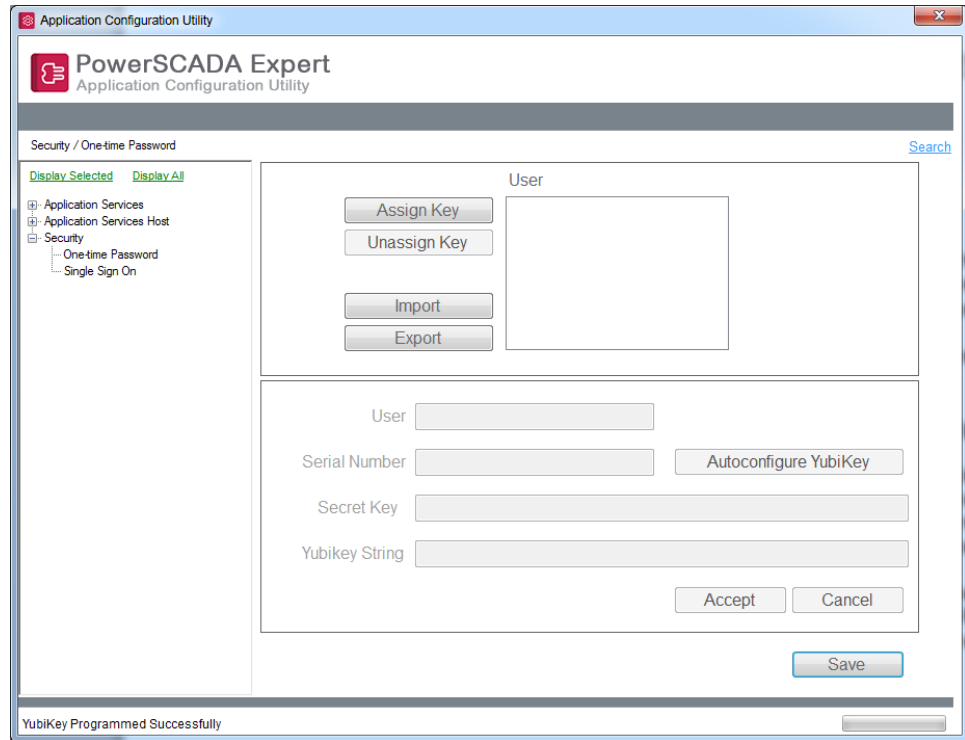
1. Under Output Settings, do not enable any of the tab buttons (when enabled, a button turns blue). Click *Enter* to enable it.

This causes a return and an "OK" to automatically occur when you press the Yubikey as part of login in PowerSCADA Expert.

2. Ignore the remaining settings. Press *Update Settings* at the bottom right of the screen.

The key is programmed. Follow the next procedure to manually configure it at the PowerSCADA Expertcomputer:

1. Access the One-Time Password screen.



2. Click Assign Key.

The fields on the lower half of the screen become live.

3. User: Type the user name that you are adding. This should be a PowerSCADA Studio user.
4. Serial Number: Type the number that is printed on the underside of the key.
5. Secret Key: Enter the Secret Key from the YubiKey Personalization Tool (created above). Enter the secret key exactly as it was created, including all spaces. After you enter it, the key will be encrypted and will display as bullets (•••) in the future.
6. Press the button on the top of the YubiKey.
7. Yubikey String: This field is populated when you press the button in step 6.
8. Click **Accept**.
9. Repeat the steps above for any additional keys.

## Import and Export One-Time Password Settings

To copy and use one-time password settings on multiple server computers, use this procedure.

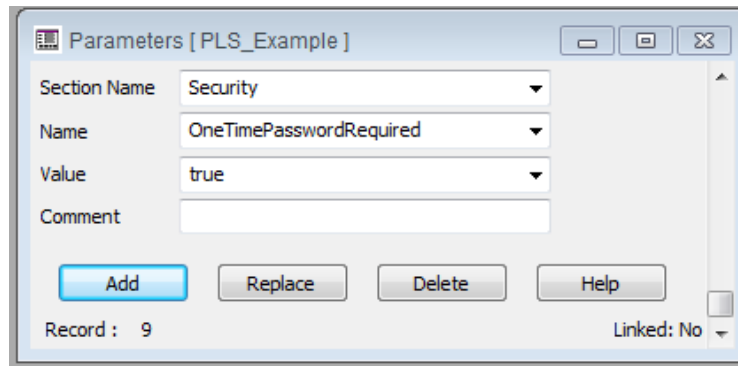
**NOTE:** When you import password settings into another server, you will overwrite any password settings that already exist there. You are not simply adding the new password settings to the existing ones.

1. On the One-time Password screen, create the password settings that you want to use.
2. Click Export. A file named *ExportedOTPCConfiguration.xml* is generated. You can rename it if you wish. Save it where you can access it from other servers, or place it on a jump drive.

- From a server to which you want to import the password settings, click Import. You are prompted for a location.
- Browse to the location where you placed the .xml file. Click Open and accept the .xml file.

## Disabling YubiKeys

To "disable" a YubiKey, use the Citect Parameter that you created for the YubiKey (see [Add the Citect Parameter on page 228](#)). Scroll to the parameter for the Yubikey. Change the Value from **true** to **false**. Click Replace, then compile the project.

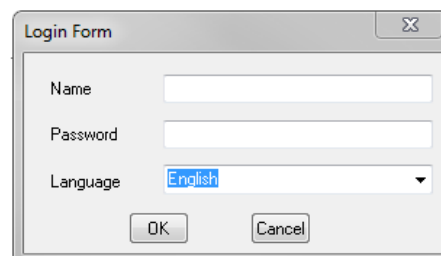


## Log in with a Programmed YubiKey and One-Time Password

After the key is programmed and associated with a user in PowerSCADA Expert, and you have enabled YubiKey usage, the user will use the key to log in to the system. To log in:

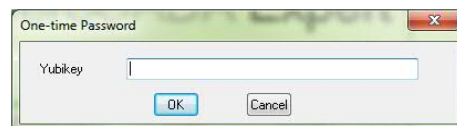
- Insert the programmed YubiKey into a USB port of the PowerSCADA Expert server.
- Launch PowerSCADA Expert Runtime, or access runtime via a remote web client.
- Run the project you want to view.
- In the upper right corner of the Startup screen, click Login.

You see the PowerSCADA Studio login screen:



- Enter your name and password. Click OK.

The One-time Password screen appears:



- Press the button on the YubiKey.

The one-time password is generated. The key and software communicate behind the scenes to verify the uniqueness of the one-time password and to click OK.

You can start using runtime screens.

### Single Sign On

Use this feature to associate a project user with either a PowerSCADA Expert (PowerSCADA) or Power Monitoring Expert (PME) username/password. When the user is logged in to runtime and accesses an external application (such as Dashboards), the SSO user password is used to authenticate with the external application.

We recommend, when you use Single Sign On, that you maintain the components on the same computer or on a secure network. If higher security is needed, use SSL (https).

## NOTICE

### LOSS OF SYSTEM CONFIDENTIALITY/INTEGRITY/AVAILABILITY


Ensure that all single sign on information is secured, either on the same computer or a secure network.

**Failure to follow these instructions can result in unauthorized access to, modification of, and unavailability of the system.**

The fields are:

- Application: From the drop-down list, choose the application (such as Dashboards, Basic Reporting, Advanced Reporting, Diagrams, LiveView) for which you want PowerSCADA project user-to-SSO user mapping.
- Timeout: The time after which the system will stop trying to find a match. If no match is found, single sign on for this user will not take place.
- Guest User: This feature allows the PowerSCADA runtime operator to access any of the integrated applications in PME or PowerSCADA Expert(basic reports), but the operator will be acting as a Guest User and will have fewer feature privileges. Check Guest User, then click Edit to launch the Edit User dialog. In the Edit User dialog, type the SSO user and password that match the username and password of the PME or PowerSCADA user to which the Guest User is mapped. For example, you could create a guest user that only has access to dashboards, and link a PME user to this account. The PowerSCADA operator could then access dashboards without logging into the PowerSCADA runtime interface.
- Users: Add/modify the users to have access to the selected application. Use this area to add users who do need to have a PowerSCADA project user account.
  - Citect User: the project username for the user logging in to the PowerSCADA runtime interface.
  - SSO User/SSO Password: The established credentials for this user, either from PowerSCADA Expert or Power Monitoring Expert.

### Single Sign-On (SSO) Calls from a Web Client

PowerSCADA Expert automatically detects calls that are made from a web client. The calls are sent to an I/O server. For this to work properly, the user needs RPC privileges for web client access. The default is False. To change this to True, access the user account through the Security link (  ) on the System menu. Change "Allow RPC" to **TRUE**.

Note: see the [Integration Parameters](#) topic for information regarding further INI parameters that must be set for Single-Sign On with integrated applications.

## 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 236](#)

[Add and Modify User Accounts on page 237](#)

[Change Access Rights on page 238](#)

### Default User Access Settings (Privileges)

These user access rights (privileges) are created to be parallel to those in Power Monitoring Expert. 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 <sup>1</sup>	Access Level Label	Access Level Value	Operator1	Operator2	Engineer	Administrator	Kernel
			2	4	6	7	8
Circuit breaker, switch control	PL_Sec_CBControl	1		X	X	X	X
IED configuration	PL_Sec_IEDConfig	2			X	X	X
Circuit breaker tagging	PL_Sec_Tagging	3			X	X	X
Alarms acknowledgment	PL_Sec_AlmAck	4	X	X	X	X	X
Alarm deletion	PL_Sec_AlmDelete	5			X	X	X
Alarm configuration	PL_Sec_AlmConfig	6			X	X	X
Add/remove log-in users	PL_Sec_UserConfig	7				X	X
Reset alarms in device	PL_Sec_AlmReset	8			X	X	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 238](#) 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
- CtExplor.exe - PowerSCADA Studio
- ProfileWizard.exe - I/O Device Manager
- 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

**NOTE:** A security parameter in Citect.ini controls whether users are required to enter a password when they initiate a breaker control in runtime. The default setting is 1 (true), meaning that all users will be forced to enter a password to operate a control, even if they are logged in. The 0 setting (false) means that no user will ever be required to enter a password when operating a control. See [Security Parameters on page 256](#) for parameter information.

The Users screen controls all of the user access levels for each project. To assign user access, follow these steps:

1. From the PowerSCADA Studio Projects page, choose the project for which you want to assign user access.
2. Click the Security link (  ) on the left hand side of the window.
3. For each 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 236](#) for the level of each type of access right.

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 link a Windows user to a PowerSCADA Expert role, you should add the "role" that specifies the Windows group of which the Windows user is a member.

The pre-existing AutoLogin capability is extended to include the client, when the user is a Windows user, having an associated PowerSCADA Expert role. In order to invoke

this functionality for a Windows user, you need to set the [Client]AutoLoginMode parameter in the Citect.ini file.

Instead of using auto-login when the system starts up, users can also log in to PowerSCADA Expert using any Windows user credential that is a member of the linked group.

When the name of a PowerSCADA Expert user also has the same name as a Windows user, the PowerSCADA Expert user takes priority at runtime. However, if a valid PowerSCADA Expert user login does not succeed for some reason, the Windows user credentials will not be checked and an alert will be generated to advise that the login was not effective.

For more information, see **Using PowerSCADA Expert with Windows Security** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## Change Access Rights

You can edit the default access rights to each of the eight levels, thus changing the privileges that are available at each level. This is done in the PLSecurity.ci file (in the PLS\_Include project).

To change access rights:

1. Open PLSecurity.ci.
2. Locate the AccessRights section.
3. For each right that you want to add (for example, to add access privileges for working with switches), add a new CASE.
4. Save and close PLSecurity.ci.

NOTE: PLSecurity.ci is overwritten any time that the PLS\_Include project is overwritten. This happens with every release of the product, including service packs. To ensure that you do not lose changes that you enter, you need to:

- Note the changes that you make. Keep a copy of these changes.
- Re-merge the changes every time that the PLS\_Include project is overwritten.

## Appendix 1: Default Genie Library

The genie library includes a number of general genies for objects such as motors and pumps. There are also genies that are specific to PowerSCADA Expert. These genies use a particular naming convention, which is described in the table below. In the PowerSCADA Expert library, each genie name begins with “pls,” and is followed by a description of the type of genie according to this table:

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

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

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

### PLS\_ALARM

This library includes genies that provide functionality to alarm displays.

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

### PLS\_ANSI\_BASE\_1 / PLS\_ANSI\_BASE\_2

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

1 = small size  
2 = large size

Genie Abbreviation	Description
sl_battery_gen	single-cell battery
sl_battery_multi	multiple-cell battery
sl_capacitor	capacitor
sl_capacitor_vari	variable capacitor
SL_Closed_HV	closed circuit breaker position
sl_closed_knife	closed knife switch

Genie Abbreviation	Description
sl_closed_lv	closed low-voltage circuit breaker
sl_conductive_path_1	conductive path 1
sl_conductive_path_2	conductive path 2
sl_conductive_path_3	conductive path 3
sl_conductor_junction	conductor junction
sl_contact_nc_closed	contact break, closed
sl_contact_nc_open	contact break, open
sl_contact_no_closed	contact make, closed
sl_contact_no_open	contact make, open
sl_contact_term	contact terminal
sl_ct	current transformer
sl_in_cb_rack	incoming, circuit breaker racked out, plug
SL_In_Rack	incoming, circuit breaker racked out, socket
sl_in_sw_head	incoming switch head
sl_inductor	inductor
sl_inductor_adjust	adjustable inductor
sl_inductor_gen	general inductor
sl_inductor_magcore	magnetic core inductor
sl_inductor_vari	variable inductor
sl_open	open symbol
sl_open_fuse_sw	open isolating fuse-switch
SL_Open_HV	open high-voltage circuit breaker
sl_open_knife	open knife-type switch
SL_Open_LV	open low-voltage circuit breaker
sl_out_cb_rack	outgoing, circuit breaker racked out, plug
SL_Out_Rack	outgoing, circuit breaker racked out, socket
sl_pb_break	push-button, break
sl_pb_make	push-button, make
sl_pb_term	push-button, terminal
sl_pt	potential transformer
sl_relay	relay
sl_resistor	resistor
sl_resistor_adjust	adjustable resistor
sl_resistor_vari	variable resistor
sl_separable_con_closed	separable connector, closed
sl_separable_con_open	separable connector, open
sl_separable_con_plug	separable connector plug
sl_separable_con_socket	separable connector socket

## PLS\_ANSI\_CB\_1 / PLS\_ANSI\_CB\_2

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

1 = small size

2 = large size

Additional definitions:

- bus = busway
- cb = circuit breaker
  
- hv = high voltage
- lv = low voltage
  
- dr = drawout
- nd = non-drawout
  
- fd = earth at bottom (feeder)
- inc = earth at top (incomer)
- nes = no earth
  
- nc = not remote control
- c = remote control

Genie Abbreviation	Description
hv_cb_bus_dr_c	high-voltage drawout circuit breaker, remote control
hv_cb_bus_dr_nc	high-voltage drawout circuit breaker, not remote
hv_cb_bus_nd_c	high-voltage non-drawout circuit breaker, not remote
hv_cb_bus_nd_nc	high-voltage non-drawout circuit breaker, not remote
hv_cb_fd_dr_c	high-voltage, drawout circuit breaker, remote control, with earth at bottom
hv_cb_fd_dr_nc	high-voltage, drawout circuit breaker, no remote control, with earth at bottom
hv_cb_fd_nd_c	high-voltage, non-drawout circuit breaker, remote control, with earth at bottom
hv_cb_fd_nd_nc	high-voltage, non-drawout circuit breaker, no remote control, with earth at bottom
hv_cb_inc_dr_c	high-voltage, drawout circuit breaker, remote control, with earth at top
hv_cb_inc_dr_nc	high-voltage, drawout circuit breaker, no remote control, with earth at top
hv_cb_inc_nd_c	high-voltage, non-drawout circuit breaker, remote control, with earth at top
hv_cb_inc_nd_nc	high-voltage, non-drawout circuit breaker, no remote control, with earth at top
hv_cb_nes_dr_c	high-voltage drawout circuit breaker, remote control, no earth
hv_cb_nes_dr_nc	high-voltage drawout circuit breaker, no remote control, no earth
hv_cb_nes_nd_c	high-voltage non-drawout circuit breaker, remote control, no earth
hv_cb_nes_nd_nc	high-voltage non-drawout circuit breaker, no remote control, no earth
lv_cb_bus_dr_c	low-voltage drawout circuit breaker, remote control, busbar-type with earth at bottom

Genie Abbreviation	Description
lv_cb_bus_dr_nc	low-voltage drawout circuit breaker, no remote control, busbar-type with earth at bottom
lv_cb_bus_nd_c	low-voltage non-drawout circuit breaker, remote control, busbar-type with earth at bottom
lv_cb_bus_nd_nc	low-voltage non-drawout circuit breaker, no remote control, busbar-type with earth at bottom
lv_cb_fd_dr_c	low-voltage drawout circuit breaker, remote control, earth on load side (bottom of drawing)
lv_cb_fd_dr_nc	low-voltage drawout circuit breaker, no remote control, earth on load side (bottom of drawing)
lv_cb_fd_nd_c	low-voltage non-drawout circuit breaker, remote control, earth on load side (bottom of drawing)
lv_cb_fd_nd_nc	low-voltage non-drawout circuit breaker, no remote control, earth on load side (bottom of drawing)
lv_cb_inc_dr_c	low-voltage drawout circuit breaker, remote control, earth on feeder (top of drawing)
lv_cb_inc_dr_nc	low-voltage drawout circuit breaker, no remote control, earth on feeder (top of drawing)
lv_cb_inc_nd_c	low-voltage non-drawout circuit breaker, remote control, earth on feeder (top of drawing)
lv_cb_inc_nd_nc	low-voltage non-drawout circuit breaker, no remote control, earth on feeder (top of drawing)
lv_cb_nes_dr_c	low voltage drawout circuit breaker, no earth, remote control
lv_cb_nes_dr_nc	low voltage drawout circuit breaker, no earth, no remote control
lv_cb_nes_nd_c	low voltage non-drawout circuit breaker, no earth, remote control
lv_cb_nes_nd_nc	low voltage non-drawout circuit breaker, no earth, no remote control

## PLS\_ANSI\_SW\_1 / PLS\_ANSI\_SW\_2

These libraries include ANSI-style switches:

1 = small size

2 = large size

Genie Abbreviation	Description
sw_fused	switch: feeder, fused
sw_fused_isolated	switch: feeder, fused, isolated
sw_general	switch: feeder, general
sw_knife	switch: knife type

## PLS\_DISPLAY

This library includes two genies that provide data row items for equipment.

Genie Abbreviation	Description
equiplistitem	data row for the equipment tag list
EquipValueItem	data row for the equipment popup

## PLS\_GEN\_BASE\_1 / PLS\_GEN\_BASE\_2

These libraries include a variety of "parts" related to generators, motors, and transformers.

Genie Abbreviation	Description
chassis_ground	chassis ground
Dev_Base	device base
es_inc	earth switch, incomer
es_out	earth switch, feeder
Gen_1	generator, option 1
Gen_2	generator, option 2
gen_AC	generator: AC
gen_DC	generator: DC
genset	engine-generator
ground	ground
Motor_1	motor, option 1
Motor_2	motor, option 2
motor_ac	motor: AC
motor_dc	motor: DC
motor_synch	motor: synchronous
SL_Base	circuit breaker base symbol
sl_br_in	circuit breaker line in, non-drawout
sl_br_out	circuit breaker line out, non-drawout
SL_Bustie	bus tie
SL_CommLoss	comms loss
SL_Discrepancy	position discrepancy
sl_harmonic_filter_1	harmonic filter 1
sl_harmonic_filter_2	harmonic filter 2
SL_In	incoming bus
SL_Local	local, rather than remote control
SL_Out	feeder
SL_Tripped	tripped
Test_CB_Control	health test for the circuit breaker control
transformer_1_in	transformer 1: general, on-line
transformer_1_in_y	transformer 1: star (wye), on-line
transformer_1_out	transformer 1: general, off-line
transformer_1_out_d	transformer 1: delta, off-line
transformer_1_out_y	transformer 1: star (wye), off-line
transformer_2_in	transformer 2: general, on-line

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

## PLS\_GEN\_CMD\_1 / PLS\_GEN\_CMD\_2

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

1 = small size

2 = large size

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

## PLS\_GEN\_EQ\_1 / PLS\_GEN\_EQ\_2

These libraries include the general equipment used to make up generators, motors, and transformers:

1 = small size

2 = large size

Genie Abbreviation	Description
busbar_horz	horizontal busbar
busbar_vert	vertical busbar
gen_ac	generator: AC
gen_dc	generator: DC
gen_nd_1	generator 1: no current designation
gen_nd_2	generator 2: no current designation
mot_ac	motor: AC
mot_dc	motor: DC
mot_nd_1	motor 1: no current designation
mot_nd_2	motor 2: no current designation
mot_syn	motor, synchronous
trans_nd_1	transformer 1: no connection designation
trans_nd_2	transformer 2: no connection designation
trans_sd_1	transformer 1: star-delta (wye-delta)
trans_sd_2	transformer 2: star delta (wye-delta)
trans_ss_1	transformer 1: star-star (wye-wye)
trans_ss_2	transformer 2: star-star (wye-wye)



## PLS\_IEC\_BASE\_1 / PLS\_IEC\_BASE\_2

These libraries include a variety of symbols for IEC equipment:

1 = small size

2 = large size

Genie Abbreviation	Description
sl_cap_bank_tuned_3	capacitor bank 3: tuned
sl_cap_bank_tuned_4	capacitor bank 4: tuned
sl_capacitor	capacitor
sl_capacitor_vari	capacitor, variable
sl_closed	closed switch
sl_contact_nc	contact break
sl_ct	contact
sl_fuse_1	fuse, option 1
sl_fuse_2	fuse, option 2
SL_Head	head
sl_head_2	head
sl_in_cb_rack	incoming, circuit breaker when racked out, plug
SL_In_Rack	incoming, circuit breaker when racked out, socket
sl_in_sw_hd_isol	incoming, switch head, isolated
sl_in_sw_head	incoming, switch head
sl_inductor	inductor
sl_inductor_adjust	inductor, adjustable
SL_Open	open
sl_out_cb_rack	feeder, circuit breaker when racked out, plug
SL_Out_Rack	feeder, circuit breaker when racked out, socket
sl_resistor	resistor
sl_resistor_adjust	resistor with adjustable contact
sl_resistor_vari	resistor, variable
sl_sw_static_1	static switch 1
sl_sw_static_2	static switch 2

## PLS\_IEC\_CB\_1 / PLS\_IEC\_CB\_2

These libraries include high-voltage drawout circuit breakers:

1 = small size

2 = large size

Genie Abbreviation	Description
hv_cb_bus_dr_c	high-voltage drawout circuit breaker, remote control
hv_cb_bus_dr_nc	high-voltage drawout circuit breaker, no remote control
hv_cb_bus_nd_c	high-voltage non-drawout circuit breaker, remote control

Genie Abbreviation	Description
hv_cb_bus_nd_nc	high-voltage non-drawout circuit breaker, no remote control
hv_cb_fd_dr_c	high-voltage, drawout circuit breaker, remote control, earth at bottom
hv_cb_fd_dr_nc	high-voltage, drawout circuit breaker, no remote control, earth at bottom
hv_cb_fd_nd_c	high-voltage, non-drawout circuit breaker, remote control, earth at bottom
hv_cb_fd_nd_nc	high-voltage, non-drawout circuit breaker, no remote control, earth at bottom
hv_cb_inc_dr_c	high-voltage, drawout circuit breaker, remote control, earth at top
hv_cb_inc_dr_nc	high-voltage, drawout circuit breaker, no remote control, earth at top
hv_cb_inc_nd_c	high-voltage, non-drawout circuit breaker, remote control, earth at top
hv_cb_inc_nd_nc	high-voltage, non-drawout circuit breaker, no remote control, earth at top
hv_cb_nes_dr_c	high-voltage drawout circuit breaker, remote control, no earth
hv_cb_nes_dr_nc	high-voltage drawout circuit breaker, no remote control, no earth
hv_cb_nes_nd_c	high-voltage non-drawout circuit breaker, remote control, no earth
hv_cb_nes_nd_nc	high-voltage non-drawout circuit breaker, no remote control, no earth

## PLS\_IEC\_SW\_1 / PLS\_IEC\_SW\_2

These libraries include IEC-style switches:

1 = small size

2 = large size

Genie Abbreviation	Description
sw_general	general switch
sw_isolated	isolated switch

## PLS\_METER

This library includes meter symbols.

Genie Abbreviation	Description
circuit_monitor	PowerSCADA Expert circuit monitor
egx	PowerSCADA Expert EGX
generic_meter	generic meter
ion_7650	ION 7650 meter
micrologic	all Mircologic meters
power_meter	PowerSCADA Expert power meter

Genie Abbreviation	Description
quantum	PowerSCADA Expert Quantum
sepam	all Sepam metersI

## ITEM1

This library includes miscellaneous symbols.

Genie Abbreviation	Description
Item1	value type and units block for a circuit breaker
Item2	value type and units block for a circuit breaker
tab1	menu tab
Tab2	menu tab

## Appendix 2: Citect INI Parameters

There are a number of Citect INI parameters that you may use to configure driver parameters. These settings may be configured at the protocol level, cluster level, port level, or device level. More specific settings will override a general one. The order of precedence is:

Protocol Name > Cluster Name > Port Name > I/O Device Name

The level at which you want the INI settings to be in effect determines the name you define. For example:

To set the default timeout for all devices using the Micrologic protocol, use:

**[MICROLOGIC]  
Timeout = 2000**

To override this default for cluster 'Cluster\_1,' use:

**[MICROLOGIC.Cluster\_1]  
Timeout = 1000**

To override the default value for port 'Port\_1' on cluster 'Cluster\_1,' use:

**[MICROLOGIC.Cluster\_1.Port\_1]  
Timeout = 3000**

To override the default value for I/O device 'CircuitBreaker\_1' on port 'Port\_1' on cluster 'Cluster\_1,' use:

**[MICROLOGIC.Cluster\_1.Port\_1.CircuitBreaker\_1]  
Timeout = 4000**

Most settings can be configured to be specific to a particular I/O device. Exceptions are noted in the description for the individual parameter.

## Parameters Database

All INI parameters described in the sections below can be set in the Parameters database. Using special syntax, you can access the parameters from the PowerSCADA Studio (click Setup, then choose Parameters from the top toolbar).

- 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 249](#)

[Performance Tuning Parameters on page 256](#)

[Waveform Parameters on page 263](#)

[Sepam Event Reading Parameters on page 265](#)

[MicroLogic Modules Configuration Parameters on page 266](#)

[Data Replication Parameters on page 267](#)

## General PowerSCADA Expert Parameters

The following parameters are common to all PowerSCADA Expert devices.

### watchtime

Controls how often the product will interrogate the driver to determine whether it is still online. This parameter can only be configured for an entire driver, and hence will have the driver dll name as its section name. Where another setting may be [PM870], to set this setting it must be [PLOGIC], as PLOGIC is the name of the dll. This is the only parameter whose section name is defined in this fashion.

Parameter type: seconds

Default Value: 2

Example: [SEPAM] watchtime = 5

### kernelStatisticUpdateRate

Controls how frequently the statistics displayed in the driver kernel window are updated. This time period can be increased in order to decrease CPU load. This parameter can only be configured for the entire protocol (as with the watchtime parameter); it will have the driver dll name as its section name.

Parameter type: milliseconds

Default value: 5000

Examples:

[SEPAM40]

kernelStatisticUpdateRate = 20000

[SEPAM80]

kernelStatisticUpdateRate = 10000

### UseWriteMultiRegistersOnly

Controls PWRMODBUS driver behaviour when a single register is to be written. This parameter is set to 1 by default, enabling all writes to be made using "write multiple registers" MODBUS function. Setting this parameter to 0 allows driver to perform write using "write single register" function if (and only if) one MODBUS register is about to be written in current operation.

Parameter type: integer

Default value: 1

Examples:

[PWRMODBUS]

UseWriteMultiRegistersOnly = 1

[PWRMODBUS.MYCLUSTER.PORT\_1.BCM1]

UseWriteMultiRegistersOnly = 0

## timeout

Controls how long the driver waits for a response from a device before setting that device as offline. This value should be greater than the device/gateway timeout period. A timed out request will not be retried. The reason for this is that TCP is a guaranteed transport mechanism, and the lack of a response indicates that the device is offline or communication has been lost with that device. A device connected via a gateway should use the gateway's retry mechanism.

Parameter type: milliseconds

Default value: 5000

Examples:

```
[SEPAM40]
```

```
Timeout = 2000
```

```
[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]
```

```
Timeout = 15000
```

## retry

Defines the number of retry attempts for specific MODBUS requests. Retries may occur either when the request is timed out or certain MODBUS exception reply messages are received. The exact behaviour is controlled by the RetryTimeout and RetryException parameters.

Parameter type: number of attempts

Default value: 3

Examples:

```
[SEPAM40]
```

```
retry = 1
```

```
[SEPAM40.MYCLUSTER.PORT_1.SEPAM_DEVICE]
```

```
retry = 5
```

## RetryTimeout

When enabled (by default), the driver will re-try a timed-out MODBUS request.

Parameter type: long (boolean)

Default value: 1

Examples:

```
[SEPAM40]
```

```
RetryTimeout = 1
```

```
[SEPAM40.MYCLUSTER.PORT_2.SEPAM_DEVICE]
```

```
RetryTimeout = 0
```

## RetryException

When enabled (disabled by default), the driver will re-try a MODBUS request that has received MODBUS Exception messages. The number of retries is defined by the Retry parameter.

When Retry Exception is enabled, retry occurs when any of the following MODBUS exception messages is received:

SLAVE\_DEVICE\_FAILURE\_EXCEPTION = 0x5  
GATEWAY\_PATH\_UNAVAILABLE\_EXCEPTION = 0xA  
GATEWAY\_TARGET\_DEVICE\_FAILED\_TO\_RESPOND\_EXCEPTION = 0xB  
SLAVE\_DEVICE\_BUSY\_EXCEPTION = 0x6  
MEMORY\_PARITY\_ERROR\_EXCEPTION = 0x8  
NEGATIVE\_ACKNOWLEDGE\_EXCPETION = 0x7

Parameter type: long (boolean)

Default value: 0

Examples:

[SEPAM40]

RetryTimeout = 1

[SEPAM40.MYCLUSTER.PORT\_2.SEPAM\_DEVICE]

RetryTimeout = 0

## standbyRefreshRate

Controls how often a standby IO server attempts to poll a device to update its cache. This time period determines the maximum age that values may be when switching from a primary IO server to a standby. Decreasing this value degrades communications to the device.

Parameter type: seconds

Default value: 60

Examples:

[SEPAM40]

standbyRefreshRate = 30

[SEPAM40.MYCLUSTER.PORT\_1.SLOW\_SEPAM]

standbyRefreshRate = 120

## standbyCheckTime

Controls how often the driver will enquire of PowerSCADA Expert as to whether it is in standby or primary mode. This value can be increased to reduce CPU load.

Parameter type: milliseconds

Default value: 500

Examples:

[SEPAM40]

standbyCheckTime = 500

[SEPAM40.MYCLUSTER.PORT\_1.SLOW\_SEPAM]

standbyCheckTime = 1000

## statusUnitCheckTime

This parameter defines how frequently the driver will try to re-establish the connection with a device that has gone offline on a port that is not disconnected. It sets the maximum rate at which the driver enquires of the device, to determine if it is still

operational. If the "watchtime" parameter is set to a longer time, that value will be used instead.

NOTE: If a network gateway has multiple devices connected to it, and one device is disconnected, the driver takes it offline and does not try to reconnect it according to this parameter's schedule. If the port is taken offline and then is reconnected, the driver will reconnect the devices immediately.

Parameter type: seconds

Default value: 5 (20 for MicroLogic)

Example:

```
[SEPAM40]
statusUnitCheckTime = 5
[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]
standbyCheckTime = 10
```

## initUnitCheckTime

Controls how long the driver waits before attempting to bring a device online after it has gone offline. This value can be decreased to bring offline devices back into service in a shorter period of time. In a multi-drop scenario, this time should be relatively long, to prevent init unit requests from stalling communications to the rest of the devices on that port.

Parameter type: seconds

Default value: 120

Examples:

```
[SEPAM40]
initUnitCheckTime = 5
[SEPAM40.MYCLUSTER.PORT_1]
initUnitCheckTime = 120
```

## initCacheTimeout

Controls how long the driver will spend attempting to populate the cache before bringing a device online. When a tag has been incorrectly configured, the device will come online after this period of time.

Parameter type: seconds

Default value: 60

Examples:

```
[SEPAM40]
initCacheTimeout = 60
[SEPAM40.MYCLUSTER.PORT_1.SLOW_SEPAM]
initCacheTimeout = 30
```

## cacheRefreshTime

Controls the maximum rate at which the driver will attempt to repopulate its cache. If the driver cannot refresh its cache within the time period specified, it will collect data as fast as the network allows.



Parameter type: milliseconds

Default value: 500

Examples:

[SEPAM40]

cacheRefreshTime = 1000

[SEPAM40.MYCLUSTER.PORT\_1.FAST\_SEPAM]

cacheRefreshTime = 200

[SEPAM40.MYCLUSTER.PORT\_1.UNIMPORTANT\_DEVICE]

cacheRefreshTime = 5000

## TimeSync

Enables/disables time synchronization for the PM5000S driver. On startup and on a 15-minute schedule, the driver reads each device clock. If a device clock is not within the specified 10-second drift, the driver sets the time on that device to the current system time.

Parameter type: boolean

Default value: 0 (PM5000S) or 1 (PM5000S1)

NOTE: This is a driver-level parameter, not a protocol-level parameter. All entries must be under the PM5000S section of the .ini file. By default, the PM5000S1 protocol enables time sync. For the PM5000S, it is disabled by default because most devices will have battery backup and GPS time sync availability.

Example:

[PM5000S] TimeSync = 1

## StatusRegister

Defines a holding register that the driver reads to determine whether a device is responding to communication requests. The result of this read is not important, however it must be a valid register address within the device.

Parameter type: register address

Default value: 1100 (2 for Sepam) (PM1200 requires that this value be set to 3911)

Examples:

[PWRMODBUS]

statusRegister = 1000

[PWRMODBUS.MYCLUSTER.PORT\_DEVICE\_PM1200]

statusRegister = 3911

## StatusRegistersCount

Defines the number of registers that the driver reads to determine whether a device is responding to communication requests. The result of this read is not important, however it must be a valid register address within the device.

Parameter type: number of registers

Default value: 1 (PM1200 requires that this value be set to 2)

Examples:

[PWRMODBUS]

```
statusRegistersCount = 2
[PWRMODBUS.MYCLUSTER.PORT_DEVICE_PM1200]
statusRegistersCount = 2
```

## StatusRegisterType

Used together with StatusRegister; defines the type of the status register. Can only be configured for the PWRMODBUS driver. This parameter can have one of the following values:

- 0 - HOLDING register (default)
- 1 - INPUT register
- 2 - COIL register
- 3 - DIGITAL input (input coil) register

Any other value equals the default.

Parameter type: register type

Default value: 0

Example:

```
[PWRMODBUS]
statusRegister = 1000
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]
statusRegister = 16000
statusRegisterType = 2
```

## ModbusBase

Defines the base address for a device. Some MODBUS device registers are defined using a base address of 1. In this case, reading register 100 would actually require reading register 99. In other devices (such as the Sepam) the base address is 0. This parameter allows the base address to be configured according to the device.

Parameter type: integer

Default value: 0 for Sepam; 1 for all other drivers

Examples:

```
[PWRMODBUS]
ModbusBase = 1
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]
ModbusBase = 0
```

## RegMode

Specifies the order of bytes in a device register. It can only be set for PWRMODBUS driver, and is supposed to be unit-specific. Value values are:

	RegMode	Order of bytes
Big endian (default)	0	1 0
Little endian	1	0 1

Any other value reverts to big endian.

Parameter type: integer

Default value: 0

Examples:

```
[PWRMODBUS]
```

```
RrMode = 0 # Default
```

```
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_A]
```

```
RegMode = 1 # This device has little endian registers
```

## timeZone

Time zone names are taken directly from the Windows registry database (case-insensitive), and will otherwise default to using the I/O server's local time zone. The Windows time zone database is located in the Windows registry in:

HKEY\_LOCAL\_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\Time Zones\.

The examples of time zone names are:

AUS Central Standard Time

China Standard Time

Pacific Standard Time (Mexico)

Use the general section [POWERLOGICCORE] to specify the time zone for all devices, e.g.

```
[POWERLOGICCORE]
```

```
Timezone = Mountain Standard Time
```

This sets the default time zone for all devices (Sepam, PLogic, Micrologic, PWRMODBUS). Otherwise the time zone can be specified for each device with precedence taken as described in the start of this section.

Examples:

```
[PLOGIC870.Cluster1.Singapore_Port]
```

```
Timezone = Singapore Standard Time
```

```
[PLOGIC870.Aus_Cluster]
```

```
Timezone = Aus Central Standard Time
```

Not having a time zone specification means that the device is in the same time zone as the machine where the I/O Server is running. No time conversion will be done.

## Alarm Parameters

The following parameters are used for alarms.

### UsePLSFilter

Controls whether alarm/event filtering is done by the PLSCADA filter form or the PowerSCADA Studio 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

## Security Parameters

Use the following security parameters to add system security

### EnterPasswordForControl

This parameter controls whether users must enter a password when they control a breaker. Regardless of whether the user is logged in, a setting of 1 (true) will require a password when the user initiates breaker control. When set to 0 (false), the password check is removed. In this case, no user will be required to enter a password to control a breaker.

Parameter type: integer

Default: 1 (true)

## 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
```

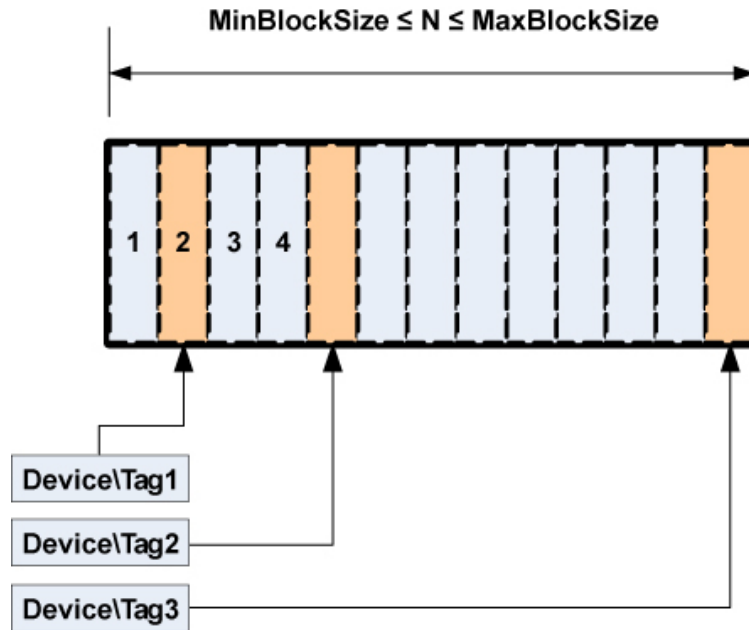
```
[PWRMODBUS.MYCLUSTER.PORT_1.DEVICE_B]
```

```
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 as “invalid memory access blocks.”

These devices create a challenge for the PWRMODBUS driver. If the device has invalid blocks that do not support scattered reads (or they are disabled for this device), the driver may try to read registers in blocks that intersect with the registers that cannot be read. This can result in the whole block being invalidated and, in certain cases, may also result in device being taken offline. Figure 1 (below) illustrates an invalid block in the middle of an address space.

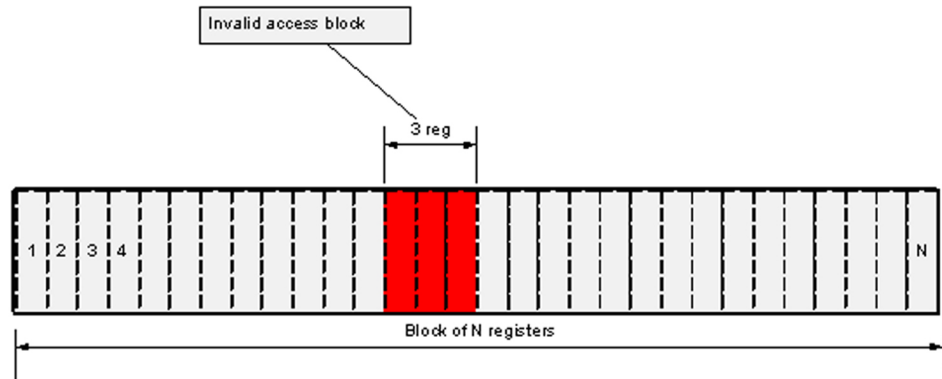
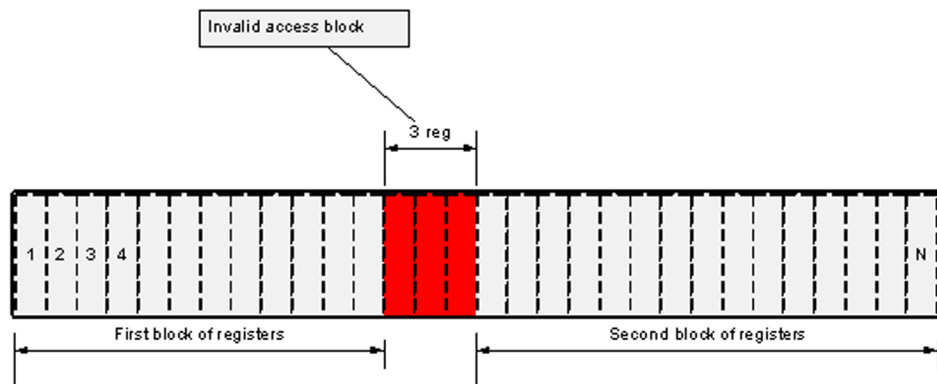


Figure 1 shows how the "invalid memory access block" affects MODBUS register blocking. In this situation, if the driver does not know that the block of 3 inaccessible registers exists, it will try to block all registers from 1 to N (depending on data that was requested by the real-time data collector). This block, however, will never be read successfully, as the device will respond with an exception to all attempts to read invalid registers.

If the configuration includes information about invalid memory access blocks, the driver will create two blocks instead of one, as shown in Figure 2:



In Figure 2, invalid registers were taken into account when the block was constructed. When configuring device that has invalid memory areas, it is especially important to define all blocks that may interfere with any of the tags.

**Invalid Block Tag Definition Syntax**

Invalid access memory areas are defined as variable tags, using the following format for the address:

```
T:IB;{m|i|c|s}:<start_register>;u<count>;E:1;L:P:0
```

where

- m, i, c and s define the type of MODBUS register
- <start\_register> is the first register address of the invalid access block
- <count> defines the number of registers in the invalid access block

Example:

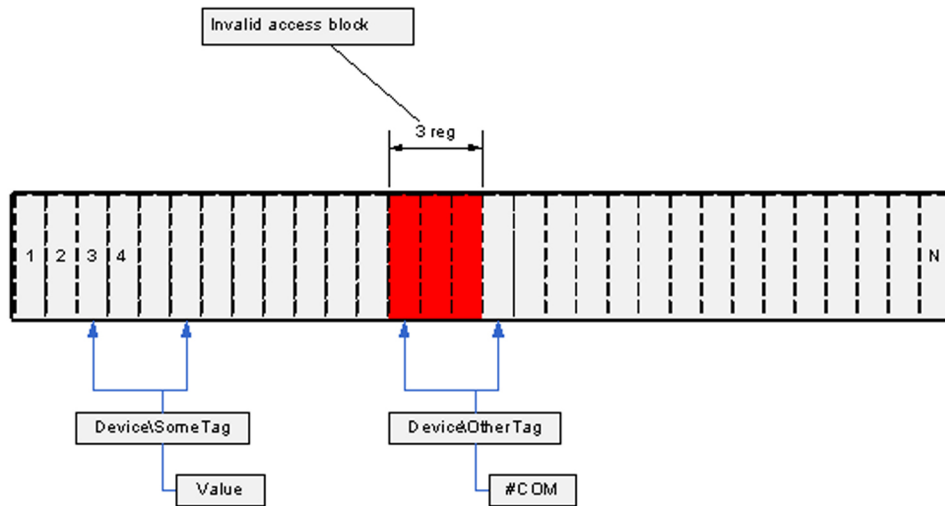
```
T:IB;m:300;u10;E:1;L:P:0
```



This defines an invalid access block of ten holding registers starting from register 300.

### Configuration Notes

When one or more invalid access blocks is defined according to the syntax above, tags configured to read any of invalid registers will be affected by it. If any of the tag registers fall into an invalid memory access block, this tag will not be readable; any attempt to read its value will result in #COM, as shown on Figure 3:



However, such tags do not affect other tags, because the PWRMODBUS driver implements algorithms that prevent tags from being invalidated by invalid memory block logic.

Tags that try to use invalid registers are detected on startup and can be found by analysing the log file. This is an example trace:

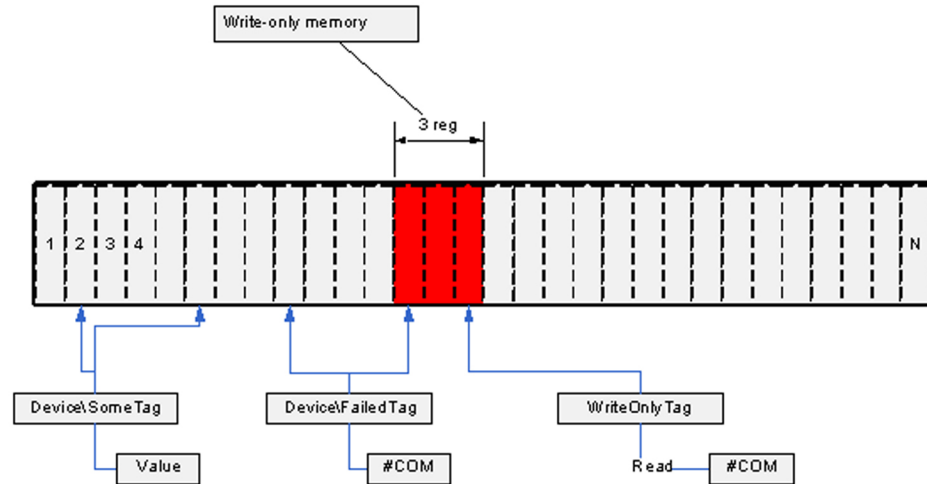
```
[DEBUG] [REAL] [GeneralDriver::BaseDatapointBuilder::BuildDataPoints
()] Adding datapoint. Tag - BCM1\H_QIVR34\SwlStr Address -
T:SS;m:283;2;E:1;L:P:26 Datapoint: class Datapoints::Status_SS
[DEBUG] [REAL] [RealTimeData::DeviceCache::Subscribe()] Init
Registers: Polled Registers: Address:283 Type:3
[ERROR] [MISC] [RealTimeData::BlockBuilder::AddDataPoint()] Cannot
add datapoint, one or more invalid memory addresses fall into non-
splittable block
[ERROR] [MISC]
[GeneralDriver::BaseDatapointBuilder::BuildDataPoints()] Could not
init datapoint. Tag BCM1\H_QIVR34\SwlStr Address
T:SS;m:283;2;E:1;L:P:26. Analyze other messages, this tag address
may contain invalid registers
```

Such output is expected when a holding register with address 283 is declared invalid. This trace helps figure out any configuration issues.

### Write-only Memory

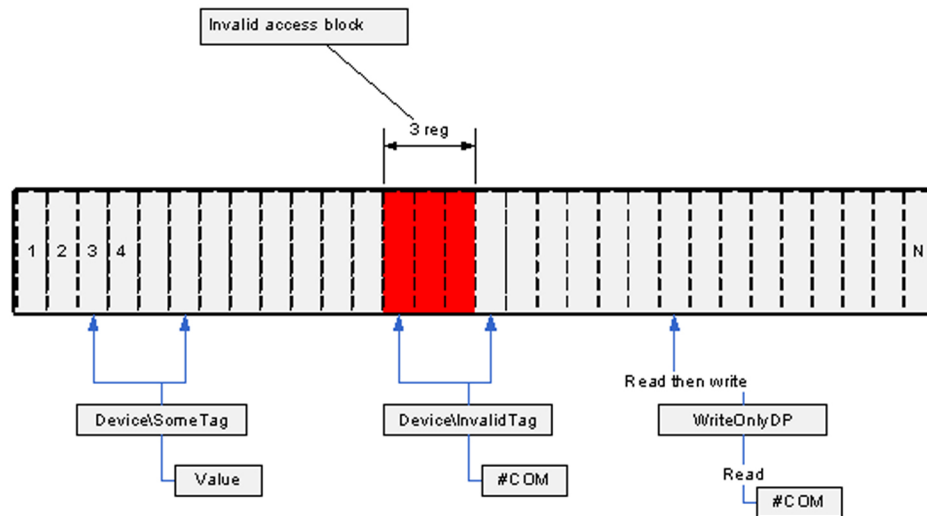
If a certain register range is accessible for write only, no additional configuration may be needed. However, to prevent the driver block optimizer from including these registers in a poll, they also must be configured by adding "invalid block" tags as

described in the *Invalid Block Tag Definition Syntax*, described above. Declaring these registers invalid will not prevent drivers from trying to write to them. Figure 4 illustrates a write-only memory configuration:



Write-only registers should not be confused with write-only datapoints that internally read a register before attempting to write. Declaring the register they read invalid will result in a datapoint not working; such mistake should be avoided. Figure 4 shows "WriteOnlyDP" as an example; this tag cannot be read (it will result in #COM), but internally it needs to get the register value before writing into it. If this register was declared invalid, tag writes would also not succeed.

Figure 5 illustrates a write-only datapoint:



**Tag Blocking Notes**

The drivers support an advanced blocking mechanism for tags. That is, real-time tags are no longer blocked together with write-only tags.

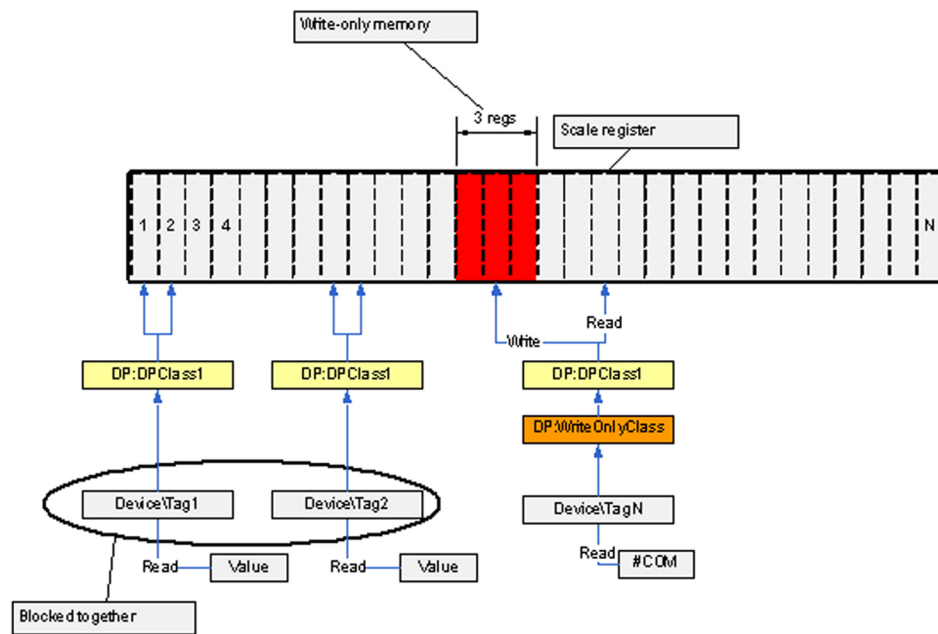
Tags found invalid, due to intersection with invalid memory areas, are not blocked with “good” real-time tags and will not therefore interfere with them.

### Write-only Tags

Beginning with driver version 2.0.1.1, the write-only tags feature is fully supported.

There are no special logic codes or address formats for write-only tags. If a tag references memory that was declared invalid (see *Invalid block tag definition syntax*, above), and its datapoint has writing capabilities, the tag becomes write only. No preliminary checks are performed to verify that the memory can be written to, and no additional configuration is needed. It is assumed that, if the tag is configured to write into memory that has been declared “invalid,” the memory can actually be written to.

It is important to understand that scaled write tags (code 110) will become write-only tags, if that scale register can be read. Tag Device\TagN on Figure 6 explains this case: the datapoint needs to read the scale value from the scale register in order to write scaled value to write-only register. However, as long as the actual register belongs to the memory that can only be written to (and it is configured using T:IB tag syntax as explained in *Invalid Block Tag Definition Syntax* above), this tag cannot be read.



The fact that the tag mentioned before cannot be read will not affect other tags reads (see *Tag Blocking Notes*, above).

Note: The write-through feature of the device cache is disabled for write-only tags.

## Waveform Parameters

The following parameters configure the waveform downloading behaviour. These parameters are only applicable for Sepam devices and PowerLogic devices that support waveforms.

[Parameter] [Default Value] [Parameter Type]

WaveformsDisable 0 Boolean value

WaveformMatchMargin 10 seconds  
WaveformCheckTime 30 seconds (PM/CM)  
WaveformZone 1 integer (Sepam)

## WaveformsDisable

This parameter enables or disables waveform downloading for a particular device.

Parameter type: Boolean value

Default value: 0

Example:

```
[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]  
WaveformsDisable 1 //Disable waveform downloading
```

NOTE: This INI setting is a global setting that sets the default at startup. You can set this for any set of devices (such as clusters, individual devices)

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

## WaveformMatchMargin

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

Parameter type: seconds

Default value: 10

Example:

```
[SEPAM40]  
WaveformMatchMargin 2
```

## WaveformCheckTime (PM, CM, and Sepam)

This parameter defines the time the driver will wait between checking for new waveforms.

Parameter type: seconds

Default value: 30

Example:

```
[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]  
WaveformCheckTime 60 (checks every 60 seconds)
```

## WaveformZone (Sepam)

This parameter defines the Sepam waveform zone that the Sepam driver will use to collect waveforms from the device. This allows two masters to extract waveforms from the same device. Valid values are 1 or 2.

Parameter type: integer

Default value: 1

Example:

```
[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]  
WaveformZone 2
```

## Sepam Event Reading Parameters

These two Sepam parameters are described below.

### EventTable

This parameter defines the Sepam event table that the Sepam driver uses to collect alarms from a device. This allows two masters to extract alarms from the same device. Valid values are 1 or 2.

Parameter type: integer

Default value: 1

Example:

```
[SEPAM40.MYCLUSTER.PORTO_1.DEVICE_A]  
EventTable 2
```

### EventIdle

This parameter defines the time that the driver will wait before requesting the next event from a Sepam device. It may be possible to reduce this value to increase the rate at which alarms can be retrieved from the device.

Parameter type: milliseconds

Default value: 500

Example:

```
[SEPAM40.MYCLUSTER.PORTO_1.DEVICE_A]  
EventIdle 200
```

## Sepam Device Driver INI Configuration Settings

Sepam devices support two event buffers, which enables two concurrent masters to read events. For all Sepam devices, the first buffer starts at register 0x40, and the second starts at register 0x70. By default, the first buffer is used; however, in certain configurations, there may be a need to tell the driver to use the second buffer. This can be done by adding the following section to citect.ini (see [Use Cicode to Customize a Project on page 159](#)):

```
[Sepam]  
[Parameter] [Default Value] [Parameter Type]  
EventTable 1 //Valid values are 1 and 2.
```

Value 2 tells the driver to use event buffer starting at 0x70; any other value falls back to 0x40.

NOTE: If the installation uses any other software—such as SMS, CET, or ION—the setting in that application should be buffer 2.

```
[Parameter] [Default Value] [Parameter Type]  
EventIdle 500 Integer
```

'EventIdle' is the time the driver will wait before requesting the next event from the Sepam device. It may be possible to reduce this value to increase the rate at which alarms can be retrieved from the device.

Example.

```
[SEPAM40.MYCLUSTER.PORT_1.DEVICE_A]
```

```
EventIdle 200
```

See [Edit Tag Addresses on page 52](#) for information about PowerLogic device driver addresses.

## 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.

### IFE/IFM

This parameter specifies whether the Micrologic device is connected through an IFE/IFM, or through the CCM (cradle comms module) or a Modbus Gateway.

0 - connection is through a Modbus Gateway

1 - connection is through an IFE/IFM

### MicrologicType

This parameter, which indicates the Micrologic Type, enables/disables functionality that can increase system performance.

1 - Type A: Only the Circuit Breaker Manager (BCM) alarm file is read.

2 - Type E: Only the Circuit Breaker Manager (BCM) file is read.

3 - Type P: The Circuit Breaker Manager (BCM) and Protection Manager (PM) alarm files are read.

4 - Type H: The Circuit Breaker Manager (BCM), Protection manager (PM), and Metering Manager (MM) alarm files are read. Waveform files are also read.

### CCM

The CCM parameter specifies whether a CCM is present on the device or if the driver should try to detect its presence ("auto mode"). Valid values are:

CCM not present - 0

CCM present - 1

Auto mode - 2 (default)

Any other value reverts to auto mode.

Parameter type: integer

Default value: 2

Example:

```
[Micrologic.MYCLUSTER.PORT_1.DEVICE_A]
```

```
CCM=1
```

## Module-Specific Packet Blocking Optimisation Settings

Due to different firmware versions, MicroLogic modules may require different blocking settings. This is especially true when MicroLogic contains a BCM that supports

MODBUS "read multiple registers" requests for up to 124 registers, and an MM or a PM module that supports 21 register reads at max. The MicroLogic driver allows blocking optimisation parameters to be overridden for each of the device's modules, as in the following example:

```
[Micrologic.MYCLUSTER.PORT_1.DEVICE_A]
```

```
maxBlockSize = 124
```

```
[Micrologic.MYCLUSTER.PORT_1.DEVICE_A.BCM]
```

```
maxBlockSize = 21
```

The parameter set for the device applies to all of its modules unless overridden in a module-specific section (e.g., [Micrologic.MYCLUSTER.PORT\_1.DEVICE\_A.BCM])

These parameters can be overridden:

- enableScatteredReads
- minBlockSize
- maxBlockSize
- PercentBlockFill

This applies to the BCM, CCM, MM, and PM modules.

## MicrologicV INI Settings

The MicrologicV device driver includes these additional INI settings:

- Level3: This is the level 3 device password (four digits), used by the driver when executing commands.
- Level4: This is the level 4 device password (four digits), used by the driver when executing commands.

If you do not supply this parameter, the driver uses the default device passwords.

## Data Replication Parameters

These parameters are used to configure the data directory paths of your servers. These settings are server wide, and must be added to the 'WaveformDB' area of the ini file.

### Database root folder path

Waveform databases for all units will locate on the filesystem under the same common folder. The path to the root folder will be specified in the citect.ini file:

```
[WaveformDB]
```

```
LocalRoot = c:\path\to\the\database\root
```

This path must be specified as local path.

By default, the PowerSCADA Expert[DATA] directory is be used as database root folder.

### Database root UNC path

For waveform files to be accessible by the remote clients, the database root folder must be available as network shared folder. The UNC name of this folder must be specified in the INI file

```
[WaveformDB]
```

```
UNCPath = \\computerName\shareToTheLocalRootAbove
```

If the UNC path to the database root is not specified, all waveform file names returned by the library will be local file names for the I/O server, making viewing the waveforms on the remote clients impossible.

## Replication destination configuration

In redundant scenario, the replication target folder must be specified for replication to work

[WaveformDB]

ReplicationDestinationRoot=\\OtherMachine\share\path

The destination path is the name of the network share on the redundant machine where its waveform database root is located. It must also allow write access.

No default value for it is assumed.

If not set or share is not accessible, no replication will be performed.

## Graphics Library Parameters

### Maximum number of entries that can be held in Event Log

The Alarm Summary length parameter in Citect.ini defines the maximum number of entries that can be held in the Event Log (default = 5000 entries). You can view all events in the Event Log and alarms in the alarm logs (Alarm Log, Unacknowledged Alarms, Disabled Alarms).

Each event requires 256 bytes of memory, plus the length of the comment. 32,000 entries will require at least 8 MB of memory. If you have many events, you should ensure that there is enough memory to store them in RAM.

After the parameter number is reached, older events are FIFO'd out to storage in [Installed Project Directory]\Schneider Electric\PowerSCADA Expert 8.2 with Advanced Reporting and Dashboards\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

**[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:  $\geq 0$   
Default Value: 0

**[General] IODevCheckInterval**

The time interval in seconds that the I/O server repeats the I/O device status check.

Allowed Values:  $\geq 2$   
Default Value: 2

## Integration and Single Sign-On Parameters

The following parameters deal with single sign-on and integration of PowerSCADA Expert, Power Monitoring Expert and Event Notification Module.

**[Applications]PSEHostName**

Required in order to use PowerSCADA Expert basic reports. This parameter specifies the IP address for the PowerSCADA Expert server.

**[Applications]HostName**

This parameter specifies the IP address for Power Monitoring Expert.

**[Applications]WebReachServer**

Default value: empty string. This parameter specifies the host name or IP address of the WebReach server machine. In most cases this is the same as the Hostname above. Required for integration with WebReach to display Diagrams in the PowerSCADA Expert runtime graphic pages.

**[Applications]PrivLevel**

Default value: 0. Allows the use of the "privilege level" field associated with PowerSCADA project users. It can be configured on a per-application level including:

PSEReporting, Advanced Reports (PME), WebReach, Dashboards, and provides the ability to limit use of Single Sign-On operations to specific privileges.

**[Applications]Area**

Default value: 0. Allows the use of the “area” field associated with PowerSCADA project users. It can be configured on a per application level including: PSEReporting, Advanced Reports(PME), WebReach, and Dashboards, and provides the ability to limit the use of Single Sign-On operations to specific areas.

**[Applications]UseHTTPS**

Default value: FALSE. Required if the Power Monitoring Expert server has SSL certificates installed and enabled.

**[Applications]PSEHostPort**

Default value: 23103. Only used for configuring Single Sign-On to PowerSCADA Expert web applications - Basic Reports and LiveView.

**[Applications]ENMServer**

Default value: localhost. Must be set in order to configure PowerSCADA Expertgraphic pages to display the Event Notification Module Configuration web page.

**[Applications]ENMPort**

Default value: 82. Must be set to '82' in order to configure PowerSCADA Expertgraphic pages to display the Event Notification Module Configuration web page.

## 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.

### **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 I/O Device Manager, 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 219](#) 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 I/O Manager 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 85](#) 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 I/O Manager, 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.

**I/O Manager**

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 I/O Manager to add the device profile into a project.

**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.

### **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 PowerSCADA Studio.

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 236 for more information.

#### **vector math**

Vector math and vector math IEEE are two logic codes. They are the result of math that use vectors, which are directed quantities.

## **Appendix 5: Frequently Asked Questions (FAQs)**

The following items provide information about topics that generate frequent questions.

## If I don't use PowerLogic drivers, how do I create device profiles?

The easiest way to do this is to:

Create a device type using a non-PowerLogic driver (like MODNET).

1. Using that device type, create a device profile.
2. You need to change the addressing of the new device type. The easiest way to do this is to copy the addressing from a known device type, and then make the necessary changes for the new device type.

## How should we manage categories and subcategories?

We recommend that each integration team decide in advance which categories and subcategories they will use. The I/O Device Manager 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 76](#) 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 I/O Device Manager?

1. You must close all open dbf files.
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 when setting up the project (such as setpoints or data type modifications) are lost when you delete the device from PowerSCADA Expert.

## What are the requirements for device names?

### Device Name:

Keep Device name  $\leq$  16 characters. Use \_ as a separator.

If you use a naming convention that incorporates location, you will be able to do filtering on alarm location.

- For example, Site\_Building\_Panel\_device would be named Sx\_Bx\_Px\_Device. (Site1\_Building1\_Panel1\_CM41 — S1\_B1\_P1\_CM41).
- The fewer levels you have, the more characters you can have in each level.

### Device Comment:

Use this field for an alias of the device name.

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

## How do I troubleshoot device communications issues?

PowerSCADA Expert drivers provide default communication settings that work with most devices. However, in cases when communication losses occur, use this checklist for finding the issues.

1. Initial checks, if the device is attached via a gateway:
  - Ensure that all communication settings are correct on the gateway and device.
  - Check the gateway timeout. A setting that is too low will cause many timeouts to occur. A setting that is too high will impact performance. We recommend a 3 second timeout, because most devices work well with this setting. Some devices may require a higher timeout (5 seconds).
2. In all communication setups (also see the driver help for parameters):
  - Ensure that the PowerSCADA Expert driver timeout is correct. We recommend that you set this to:  
gateway timeout x number of clients + padding  
Example: If the gateway timeout is 3 seconds and there are 3 clients, set the timeout in PowerSCADA Expert to 10 seconds.
  - Check the maximum block read size. Some devices do not handle large block reads well. When you lower the maximum block read size, the requests are smaller and faster. The downside is that more requests will be sent to the device, and tags will refresh more slowly.
  - Check the device to see if there are registers that it cannot read. Some devices do not allow access to all registers.  
Example: Data is in register 100-125 and 130-150. PowerSCADA Expert will perform one read from 100-150. If 126-129 do not allow reading, this packet will return an exception. Use the appropriate logic code to mark these registers as invalid reads.
  - If there are still timeout/no response issues, enable retries on exception. Some devices may not respond if they are performing other functions. In this case, a0x0A or 0x0B exception will be returned to PowerSCADA Expert, which will cause a communication loss. Enabling the "retry on exception" will re-try the request.

## How do I use MODBUS communications methods?

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

MODBUS TCP/IP: Use this for any device that can speak TCP/IP natively. This includes CM4 or PM8 devices that have an ECC card installed.

## 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 236](#). For additional information, see **Using Security** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## How do I manage projects in the PowerSCADA Studio?

Although the designer may wish to organize each project in a particular way to suit customer's needs, the following is a recommended best practice:

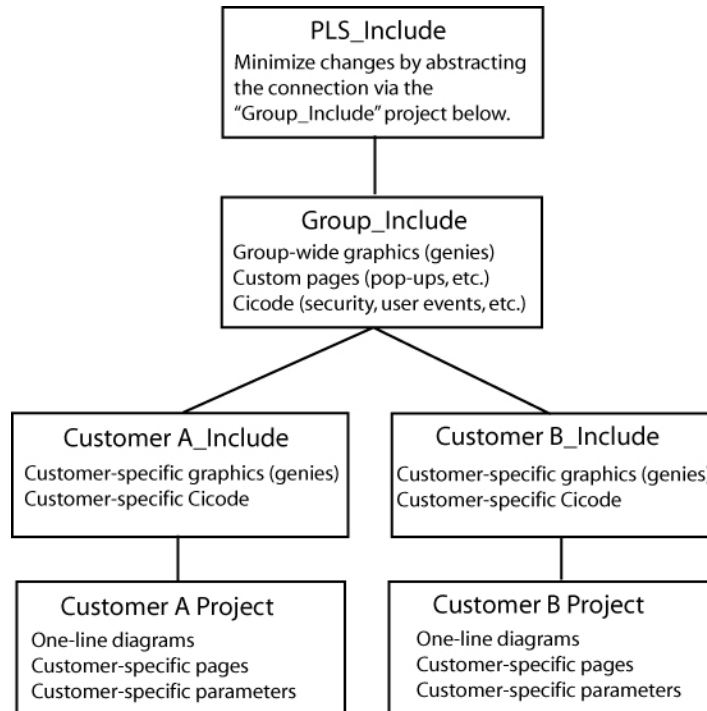
1. Keep original 'Master' copies of the PLS\_Example and the PLS\_Include projects for reference.
2. The Services group may develop a group-wide "include" project that will act as a conduit between the PLS\_Include project and all customer projects (e.g., "Group\_Include"). This will make the upgrading 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 PowerSCADA Studio 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 .frm files. Each library has its own .ctm/.frm 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 PowerSCADA Studio 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 268](#))

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 57](#), "Alarm On Text" and "Alarm Off Text").

## What can I modify during runtime?

See the *PowerSCADA Expert Design Guide*, "Updates to the System While Online," for a list of items you can modify during runtime.

## 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 111](#) for help.
- left the INI settings at <default>. In the Computer Setup Wizard, General Options Setup screen, do not change the StartupPage field; leave it as <default>.

## What can I set up in logging and archiving?

### Event Logging and Archiving:

Event fields that are logged to disk may be configured by adjusting the AlarmFormat parameter.

There is no automatic maintenance performed on the log files. It is important that the log/waveform data be cleared out periodically (to prevent the hard drive from filling up; this does not affect performance).

## How do I create and configure busbars?

When drawing one-line diagrams:

Analyse the drawings at a customer site.

Number the busbars consistently on the one line diagram(s). If busbar 14 spans across multiple pages, it should be numbered busbar 14 on all pages. Label the

voltage level (0–3) on each busbar.

#### Uses for Line Active:

**Page Connections:** Many one-line diagrams will span multiple pages. To connect these pages together, you must use the line active field of the ‘incomers’ of the second and subsequent pages. Set the line active field of the incoming busbars on these pages to an expression that references the nearest device on the same busbar of the previous page.

**Metered Busbar:** Many busbars are actually metered. It is more accurate to allow these metering devices to dictate state than to rely solely on the simulation (see Expressions below).

#### Configuration of Line Active:

**Simulation:** If the Line-Active field is left blank, the busbar state will be determined by surrounding devices.

#### Expressions:

A Cicode expression in the form of Device\Tag > Nominal Voltage (I.E., S1\_B1\_P1\_CM41\MMXU1\PhV\zavg > 120).

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

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

#### Hard-Coded:

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

## What INI parameters should I use for debugging?

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

#### Parameter: [PowerLogicCore]

DebugCategory = All

DebugLevel = All (or Error)

LogFileArchive = Deprecated; no longer used. Use *[Debug]SysLogArchive* instead.

LogFileSize = Deprecated; no longer used. Use *[Debug]SysLogSize* instead.

#### Parameter Details:

**DebugCategory** defines which message categories to log. (See table below).

**DebugLevel** defines debug levels of messages to be logged. (See table below).

#### Debug Levels

The following debug levels are accepted by PowerLogic driver core library:

WARN: log all warning level messages

ERROR: log all error messages

TRACE: log all trace messages

DEBUG: log all debug messages

ALL: include all level messages

#### Debug Categories

PowerLogic core library and drivers messages are grouped in categories. Each of these categories can be enabled independently from others in any combination.

MISC: miscellaneous messages that do not all into any other category  
ALARM: messages related to alarms, regarding collection and detection  
WAVE: messages related to waveforms -- waveforms download, processing  
REAL: messages related to real-time data collection  
STATE: messages related to internal object-state changes  
CORE: core events that do not fall into driver-specific logic  
MODBUS: TCP/MODBUS messages  
ENTRY: trace messages produced when driver API entry points are called  
PORT: traces related to the port events  
UNIT: traces related to specific unit events  
WAVETOALARM: not used  
DATAPOINT: debug messages related to data points  
STATISTICS: enables driver statistics data output  
COMMAND: messages related to commands  
REPLICATION: messages produced by replication subsystem  
ALL: enables all categories

**Parameter: [Debug]**

Menu = 1

Parameter Details:

The Menu parameter determines whether the Kernel option is displayed on the control menu of the runtime menu. This can also be enabled using the Computer Setup Editor.

## How do I tune my system for best performance?

There are several parameters that you can use to enhance your system's performance:

**Driver-tuning parameters:**

**Parameter (Back Polling Rate): [SEPAM40]**

CacheRefreshTime = 1000

InitUniCheckTime = 120

Retry = 3

Timeout = 1000

**Parameter Details:**

The CacheRefreshTime parameter controls the maximum rate at which the driver will attempt to repopulate its cache. If the driver cannot refresh its cache within the time period specified, it will collect data as fast as the network allows.

This back polling rate can be global to all devices or tuned up to a specific I/O device.

The InitUniCheckTime parameter controls how long the driver will wait before attempting to bring a device online after it has gone offline. This value can be decreased to bring offline devices back into service in a shorter period of time. In a multi-drop scenario, this time should be relatively long, to prevent init unit requests from stalling communications to the rest of the devices on that port.

The Retry parameter defines the number of retry attempts for specific MODBUS requests. Retries will only occur in response to the MODBUS errors which are

defined below.

The Timeout parameter controls how long the driver will wait for a response from a device before setting that device as offline. This value should be greater than the device/gateway timeout period.

**Parameter: [Device]**

WatchTime = 5000

**Parameter Details:**

**Device WatchTime** is the frequency that PowerSCADA Expert checks devices for history files and flushes logging data to disk.

Default: 5000

Range: 1000–3600000 milliseconds.

**Miscellaneous Parameters**

**Parameter: [Kernel]**

Task = 20000

**Parameter Details:**

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

Default Value: 256

Range: 50–32767

**Parameter: [Page]**

ScanTime = 250

**Parameter Details:**

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

Default: 250

Range: 1–60000 milliseconds

**Parameter: [ALARM]**

ScanTime = 500

**Parameter Details:**

**Alarm ScanTime** determines the rate at which alarms are scanned and processed.

Default: 500

Range: 0–60000 milliseconds

## If a tag is configured, how is it polled in the device?

In other words, is a tag only polled on demand when it is requested by a client (for example, when the operator displays a page with the tag on it)? Or are all configured tags polled all of the time, with the relative polling rates/communications bandwidth carefully managed?

The ModNet driver polls real-time tags on a user demand basis (when a user opens a page with the tags on it). Therefore, the time to retrieve data will vary, depending not only on the communications bandwidth, but on the amount of data being requested.

This can vary significantly, depending on which pages are displayed by the operators at any particular time.

The PWRMODBUS driver polls all configured tags; however, different types of tags can be polled at different relative rates, and the available communications bandwidth is carefully managed. This approach means that tag update rates are not subject to the scalability issues associated with operator actions (as is the case for the ModNet driver). It is also advantageous in that performance issues associated with communications bandwidth or IODevice response times can be determined at SAT/time of implementation and are not subject to significant change during operation.

The different tag types can be allocated relative importance in data requests, expressed as a percentage. (See Bandwidth Allocation Parameters in [Performance Tuning Parameters on page 256](#).) 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 256](#).) 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 I/O Device Manager?

No. If you change the placeholder %CLUSTER% to any other name in the I/O Device Manager, the system will be unable to find the actual cluster to which it refers.

## A device can prevent writes to its registers: how do I ensure that writes are successful?

PowerSCADA Expert cannot provide feedback about whether a write to a device register is successful. If a device is capable of preventing (blocking) writes to its registers (for example, Sepam), you need to verify that its "block" feature is not enabled. Do this at the device.

NOTE: In Cicode, you can also use the tagwrite function in blocking mode, i.e., bSync parameter = true; Check the return code: 0 = success, anything else = error. For more information, see the Cicode Programming Reference help file.

## How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes?

PowerSCADA Expert normally optimises its packets for greatest performance. This optimisation can sometimes include invalid areas of memory in devices. These invalid areas can be specifically defined and excluded from optimisation packets created by PowerSCADA Expert. See *Advanced Tag Blocking Capabilities* in [Performance Tuning Parameters](#) on page 256 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 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 PowerSCADA Studio.

## Why is my device in comms loss?

When you bring your system on line, and you find that PowerSCADA Expert has lost communications with a device, check the following:

- Verify that the physical connection is correct and secure.
- Verify the IP address.
- Verify the Modbus address.
- Check the statusRegister, statusRegistersCount, and statusRegisterType (see for details)

## How do I set up select before operate?

For systems in which you can determine that a single user is selecting a device prior to sending an open/close command, you can add a "select before operate" button.

To do this:

1. Locate the Select Before Operate tag in the variable tags.
2. Append `\str` to the end of the tag name.
3. Change the data type to STRING.
4. Click Add.

This creates the SBOw tag for the IEC 61850 advanced control screen. For more information about advanced control, see:

[Set Up IEC 61850 Advanced Control on page 116](#)

*Perform IEC 61850 Advanced Control on page 185*



## Appendix 5: Frequently Asked Questions (FAQs)

The following items provide information about topics that generate frequent questions.

### If I don't use PowerLogic drivers, how do I create device profiles?

The easiest way to do this is to:

Create a device type using a non-PowerLogic driver (like MODNET).

1. Using that device type, create a device profile.
2. You need to change the addressing of the new device type. The easiest way to do this is to copy the addressing from a known device type, and then make the necessary changes for the new device type.

### How should we manage categories and subcategories?

We recommend that each integration team decide in advance which categories and subcategories they will use. The I/O Device Manager 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 76](#) 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 I/O Device Manager?

1. You must close all open dbf files.
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 when setting up the project (such as setpoints or data type modifications) are lost when you delete the device from PowerSCADA Expert.

## What are the requirements for device names?

### Device Name:

Keep Device name  $\leq$  16 characters. Use `_` as a separator.

If you use a naming convention that incorporates location, you will be able to do filtering on alarm location.

- For example, Site\_Building\_Panel\_device would be named `Sx_Bx_Px_Device`. (`Site1_Building1_Panel1_CM41` — `S1_B1_P1_CM41`).
- The fewer levels you have, the more characters you can have in each level.

### Device Comment:

Use this field for an alias of the device name.

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

## How do I troubleshoot device communications issues?

PowerSCADA Expert drivers provide default communication settings that work with most devices. However, in cases when communication losses occur, use this checklist for finding the issues.

1. Initial checks, if the device is attached via a gateway:
  - Ensure that all communication settings are correct on the gateway and device.
  - Check the gateway timeout. A setting that is too low will cause many timeouts to occur. A setting that is too high will impact performance. We recommend a 3 second timeout, because most devices work well with this setting. Some devices may require a higher timeout (5 seconds).
2. In all communication setups (also see the driver help for parameters):
  - Ensure that the PowerSCADA Expert driver timeout is correct. We recommend that you set this to:  
gateway timeout x number of clients + padding  
Example: If the gateway timeout is 3 seconds and there are 3 clients, set the timeout in PowerSCADA Expert to 10 seconds.
  - Check the maximum block read size. Some devices do not handle large block reads well. When you lower the maximum block read size, the requests are smaller and faster. The downside is that more requests will be sent to the device, and tags will refresh more slowly.
  - Check the device to see if there are registers that it cannot read. Some devices do not allow access to all registers.  
Example: Data is in register 100-125 and 130-150. PowerSCADA Expert will perform one read from 100-150. If 126-129 do not allow reading, this packet will return an exception. Use the appropriate logic code to mark these registers as invalid reads.
  - If there are still timeout/no response issues, enable retries on exception. Some devices may not respond if they are performing other functions. In this case, `a0x0A` or `0x0B` exception will be returned to PowerSCADA Expert, which will cause a communication loss. Enabling the "retry on exception" will re-try the request.

## How do I use MODBUS communications methods?

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

MODBUS TCP/IP: Use this for any device that can speak TCP/IP natively. This includes CM4 or PM8 devices that have an ECC card installed.

## 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 236](#). For additional information, see **Using Security** in the citectSCADA.chm help file (Program Files> Schneider Electric > PowerSCADA Expert > v8.2 > bin).

## How do I manage projects in the PowerSCADA Studio?

Although the designer may wish to organize each project in a particular way to suit customer's needs, the following is a recommended best practice:

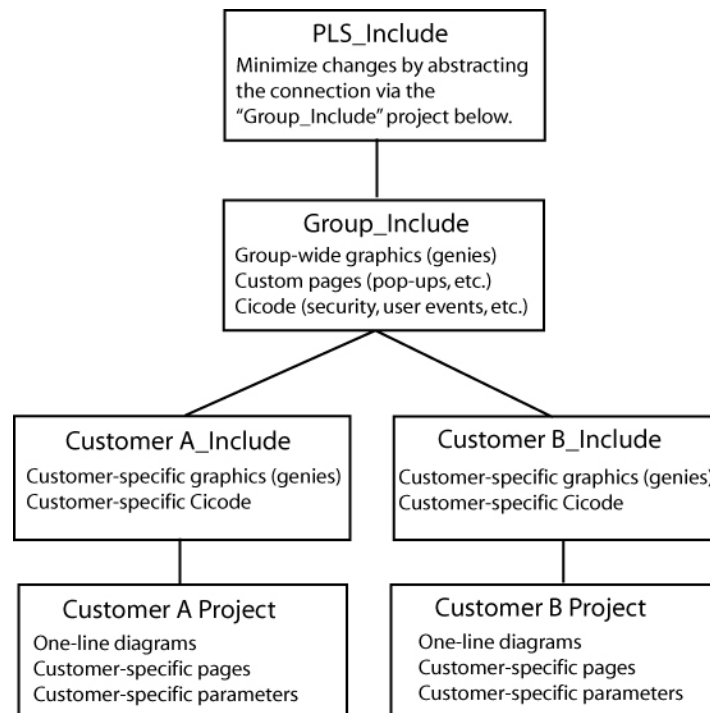
1. Keep original 'Master' copies of the PLS\_Example and the PLS\_Include projects for reference.
2. The Services group may develop a group-wide "include" project that will act as a conduit between the PLS\_Include project and all customer projects (e.g., "Group\_Include"). This will make the upgrading 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 PowerSCADA Studio 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 .frm files. Each library has its own .ctm/.frm 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 PowerSCADA Studio 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 268](#))

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 57](#), "Alarm On Text" and "Alarm Off Text").

## What can I modify during runtime?

See the *PowerSCADA Expert Design Guide*, "Updates to the System While Online," for a list of items you can modify during runtime.

## 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 111](#) for help.
- left the INI settings at <default>. In the Computer Setup Wizard, General Options Setup screen, do not change the StartupPage field; leave it as <default>.

## What can I set up in logging and archiving?

### Event Logging and Archiving:

Event fields that are logged to disk may be configured by adjusting the AlarmFormat parameter.

There is no automatic maintenance performed on the log files. It is important that the log/waveform data be cleared out periodically (to prevent the hard drive from filling up; this does not affect performance).

## How do I create and configure busbars?

When drawing one-line diagrams:

Analyse the drawings at a customer site.

Number the busbars consistently on the one line diagram(s). If busbar 14 spans across multiple pages, it should be numbered busbar 14 on all pages. Label the voltage level (0–3) on each busbar.

**Uses for Line Active:**

**Page Connections:** Many one-line diagrams will span multiple pages. To connect these pages together, you must use the line active field of the ‘incomers’ of the second and subsequent pages. Set the line active field of the incoming busbars on these pages to an expression that references the nearest device on the same busbar of the previous page.

**Metered Busbar:** Many busbars are actually metered. It is more accurate to allow these metering devices to dictate state than to rely solely on the simulation (see Expressions below).

**Configuration of Line Active:**

**Simulation:** If the Line-Active field is left blank, the busbar state will be determined by surrounding devices.

**Expressions:**

A Cicode expression in the form of Device\Tag > Nominal Voltage (I.E., S1\_B1\_P1\_CM41\MMXU1\PhV\zavg > 120).

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

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

**Hard-Coded:**

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

## What INI parameters should I use for debugging?

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

**Parameter: [PowerLogicCore]**

DebugCategory = All

DebugLevel = All (or Error)

LogFileArchive = Deprecated; no longer used. Use *[Debug]SysLogArchive* instead.

LogFileSize = Deprecated; no longer used. Use *[Debug]SysLogSize* instead.

Parameter Details:

**DebugCategory** defines which message categories to log. (See table below).

**DebugLevel** defines debug levels of messages to be logged. (See table below).

**Debug Levels**

The following debug levels are accepted by PowerLogic driver core library:

WARN: log all warning level messages

ERROR: log all error messages

TRACE: log all trace messages

DEBUG: log all debug messages

ALL: include all level messages

**Debug Categories**

PowerLogic core library and drivers messages are grouped in categories. Each of these categories can be enabled independently from others in any combination.

MISC: miscellaneous messages that do not all into any other category

ALARM: messages related to alarms, regarding collection and detection

WAVE: messages related to waveforms -- waveforms download, processing

REAL: messages related to real-time data collection

STATE: messages related to internal object-state changes

CORE: core events that do not fall into driver-specific logic

MODBUS: TCP/MODBUS messages

ENTRY: trace messages produced when driver API entry points are called

PORT: traces related to the port events

UNIT: traces related to specific unit events

WAVETOALARM: not used

DATAPPOINT: debug messages related to data points

STATISTICS: enables driver statistics data output

COMMAND: messages related to commands

REPLICATION: messages produced by replication subsystem

ALL: enables all categories

**Parameter: [Debug]**

Menu = 1

Parameter Details:

The Menu parameter determines whether the Kernel option is displayed on the control menu of the runtime menu. This can also be enabled using the Computer Setup Editor.

## How do I tune my system for best performance?

There are several parameters that you can use to enhance your system's performance:

**Driver-tuning parameters:**

**Parameter (Back Polling Rate): [SEPAM40]**

CacheRefreshTime = 1000

InitUniCheckTime = 120

Retry = 3

Timeout = 1000

**Parameter Details:**

The CacheRefreshTime parameter controls the maximum rate at which the driver will attempt to repopulate its cache. If the driver cannot refresh its cache within the time period specified, it will collect data as fast as the network allows.

This back polling rate can be global to all devices or tuned up to a specific I/O device.

The InitUniCheckTime parameter controls how long the driver will wait before attempting to bring a device online after it has gone offline. This value can be decreased to bring offline devices back into service in a shorter period of time. In a multi-drop scenario, this time should be relatively long, to prevent init unit requests from stalling communications to the rest of the devices on that port.



The Retry parameter defines the number of retry attempts for specific MODBUS requests. Retries will only occur in response to the MODBUS errors which are defined below.

The Timeout parameter controls how long the driver will wait for a response from a device before setting that device as offline. This value should be greater than the device/gateway timeout period.

**Parameter: [Device]**

WatchTime = 5000

**Parameter Details:**

**Device WatchTime** is the frequency that PowerSCADA Expert checks devices for history files and flushes logging data to disk.

Default: 5000

Range: 1000–3600000 milliseconds.

**Miscellaneous Parameters**

**Parameter: [Kernel]**

Task = 20000

**Parameter Details:**

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

Default Value: 256

Range: 50–32767

**Parameter: [Page]**

ScanTime = 250

**Parameter Details:**

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

Default: 250

Range: 1–60000 milliseconds

**Parameter: [ALARM]**

ScanTime = 500

**Parameter Details:**

**Alarm ScanTime** determines the rate at which alarms are scanned and processed.

Default: 500

Range: 0–60000 milliseconds

## If a tag is configured, how is it polled in the device?

In other words, is a tag only polled on demand when it is requested by a client (for example, when the operator displays a page with the tag on it)? Or are all configured tags polled all of the time, with the relative polling rates/communications bandwidth carefully managed?

The ModNet driver polls real-time tags on a user demand basis (when a user opens a page with the tags on it). Therefore, the time to retrieve data will vary, depending not

only on the communications bandwidth, but on the amount of data being requested. This can vary significantly, depending on which pages are displayed by the operators at any particular time.

The PWRMODBUS driver polls all configured tags; however, different types of tags can be polled at different relative rates, and the available communications bandwidth is carefully managed. This approach means that tag update rates are not subject to the scalability issues associated with operator actions (as is the case for the ModNet driver). It is also advantageous in that performance issues associated with communications bandwidth or IODevice response times can be determined at SAT/time of implementation and are not subject to significant change during operation.

The different tag types can be allocated relative importance in data requests, expressed as a percentage. (See Bandwidth Allocation Parameters in [Performance Tuning Parameters on page 256](#).) 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 256](#).) 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 I/O Device Manager?

No. If you change the placeholder %CLUSTER% to any other name in the I/O Device Manager, the system will be unable to find the actual cluster to which it refers.

## A device can prevent writes to its registers: how do I ensure that writes are successful?

PowerSCADA Expert cannot provide feedback about whether a write to a device register is successful. If a device is capable of preventing (blocking) writes to its registers (for example, Sepam), you need to verify that its "block" feature is not enabled. Do this at the device.

NOTE: In Cicode, you can also use the tagwrite function in blocking mode, i.e., bSync parameter = true; Check the return code: 0 = success, anything else = error. For more information, see the Cicode Programming Reference help file.

## How do I prevent PowerSCADA Expert from accidentally making invalid areas in memory available to reads and writes?

PowerSCADA Expert normally optimises its packets for greatest performance. This optimisation can sometimes include invalid areas of memory in devices. These invalid areas can be specifically defined and excluded from optimisation packets created by PowerSCADA Expert. See *Advanced Tag Blocking Capabilities* in [Performance Tuning Parameters on page 256](#) 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 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 PowerSCADA Studio.

## Why is my device in comms loss?

When you bring your system on line, and you find that PowerSCADA Expert has lost communications with a device, check the following:

- Verify that the physical connection is correct and secure.
- Verify the IP address.
- Verify the Modbus address.
- Check the statusRegister, statusRegistersCount, and statusRegisterType (see for details)

## How do I set up select before operate?

For systems in which you can determine that a single user is selecting a device prior to sending an open/close command, you can add a "select before operate" button.

To do this:

1. Locate the Select Before Operate tag in the variable tags.
2. Append `\str` to the end of the tag name.
3. Change the data type to STRING.
4. Click Add.

This creates the SBOw tag for the IEC 61850 advanced control screen. For more information about advanced control, see:

[Set Up IEC 61850 Advanced Control on page 116](#)

*Perform IEC 61850 Advanced Control on page 185*

# Index

- .aes file
  - encryption 224
- 1**
- 1.#R
  - value returned in instant trend 180
- A**
- access rights
  - adding for users 57
  - changing for users 238
  - Windows integrated users 237
- ACE969TP device type
  - used with DNP3 68
- ACE969TP module
  - used with DNP3 68
- additional sources of information 26
- address
  - correcting missing or invalid 42
  - functional 43, 71
    - defined 275
- advanced control
  - perform for IEC 61850 182
  - set up
    - IEC 61850 116
- advanced tag block capabilities
  - configuration notes 261
  - invalid block tag definition syntax 260
  - tag blocking notes 262
  - write-only memory 261
  - write-only tags 263
- AdvOneLine.csv 136
- AdvOneLine.exe 136
  - stopping and starting 139
- AdvOneLine.ini 136
- AdvOneLine.ini settings
  - one-lines 137
- AdvOneLineDebug.Bus.csv 136
- AdvOneLineStatusLog.txt 136
- alarm
  - adding an onboard tag 157
  - associated with waveforms 166
  - category 177
  - category number 177
  - changing pickup/dropout text 285, 298
  - changing the severity 156
  - codes
    - unique identifier 54
  - colors 173
  - filtering
    - custom 177, 179
  - historical logging parameters 156
  - issues 174
  - level
    - setting in Profile Editor 58
  - onboard 155
  - pc-based 154
    - setting up in Profile Editor 59
  - priority 177
  - priority number 177
  - setting up based on an enumeration 155

- states
    - not display correctly when genie dimensions wrong 141
  - startup parameters 268
  - tags
    - adding 155
  - alarm filters 176
  - alarm format code definitions 86
  - alarm log 173
    - column headings described 175
    - filtering 173, 176
    - freezing 173
    - logging data to disk 174
    - printing 174
    - problem with alarms not displaying 156, 174
    - sorting 172-173
  - Alarm On/Alarm Off text 60
  - alarm pages
    - adding to the project 155
  - Alarms/Events page
    - equipment column 172
    - filtering alarms 177
    - remove/insert/move columns 173
    - viewing 172
  - AlarmSummaryLength parameter
    - maximum number of entries in the Event Log 156
  - Allow RPC
    - YubiKey 228, 235
  - analysis page 179
  - and all REAL 87
  - animation
    - loss of communication not filtering correct 172
  - animation numbers (ANs)
    - used in objects 284, 297
  - Application Services
    - restart 220
    - setup 221
  - Applications 223
  - Area
    - ini parameter 270
  - array
    - genie properties 116
  - assigning tags to generic I/O points 46
  - assumptions
    - about PowerSCADA Expert system integrators 16
    - before system integrator can set up system 16
  - ATS
    - configuring 140
    - errors 149
    - information 148
  - audible alarms
    - setup 157
  - auto assign genie 144, 147
  - auto configure genie 144, 147
  - autoconfigure 228
- B**
- back ground color
    - changing in projects 110
  - background colors
    - changing 183

- backup
    - PowerSCADA Studio project file 158
    - Profile Editor project file 74
  - base unit and conversion
    - adding and editing 40
    - deleting 41
  - basic readings summary 201
  - basic reports
    - Internet Explorer 9 emulation 188
    - set up delivery method and email address 225
  - before you use the I/O Device Manager 117
  - block writes
    - using 98
  - BRCB
    - entering for IEC 61850 device 127
    - entering for IEC61850 device 127
  - breaker
    - configuring 141
    - errors 150
    - information 150
  - browser navigation buttons do not work 111
  - busbar
    - configuring 140
    - creating and configuring 285, 298
    - errors 149
    - information 149
  - busbars
    - two with same line active 147
- C**
- CacheRefreshTime parameter 287, 300
  - calculating disk storage 164, 179
  - CallProvider
    - PLSProviders.ci 159
  - CASE
    - changing user access rights 238
  - category/subcategory
    - adding 48
    - deleting 48
    - editing 48
    - managing 280, 293
  - changing alarm severity 156
  - circuit breaker status
    - using special tags to control 85
  - Citect parameters file 106
  - Citect platform connection 223
  - citect.ini
    - setting up multiple monitors 219
    - settings
      - one-line 137, 139
      - used to customize a project 159
  - clear cache 161
    - Live View 209
    - reporting 196
  - CLUSTER name (%CLUSTER%)
    - changing 289, 302
  - cluster setup
    - in the I/O Device Manager 118, 120, 123, 125
    - redundant I/O servers 22
  - collapse/expand all
    - Import Reconciliation screen 80



- color
  - active
    - busbars 286, 299
  - changing background in projects 110
  - changing for genies 111
- colors
  - assigning to one-lines 137
  - changing background 183
- comment line
  - add to CSV file 102
- comms loss
  - troubleshooting 291, 304
- communications loss 171
  - troubleshooting 291, 304
- Complete Import button 77, 81
- composite device type
  - creating 65
- Computer Setup Wizard
  - when adding a startup page 111
- configuration
  - one-line engine 224
- configuration tools
  - described 17
- configure
  - application services 220
- control format codes 86
- control inhibit 183
- controls
  - how they operate 43, 55
- conversions
  - to base units 39
- core service host
  - connection with PowerSCADA Studio 223
- core services
  - configure 220
- CoreServiceHost
  - restart when changing runtime projects 187
- create menu item for LiveView page 210
- creating
  - CSV file 47
  - device profile 57
  - engineering template 37
  - genie 115
  - one-line on a graphics page 111
  - PowerSCADA Expert project 109
  - project 68
- CSV file
  - create to add devices 129
  - creating, viewing, printing 36, 47
  - defining multiple devices 128, 132
- CTZ file
  - backup file 158
- custom
  - control format codes 87
  - filters
    - alarms 177, 179
  - genie
    - creating 115
  - icon 35
  - reset
    - adding 54

- symbol
    - creating 116
  - tag
    - deleting 52
    - deleting when associated with a device type 52
    - editing 51
  - custom tag
    - adding 50
  - customer support contact information 26
  - customize tag names
    - sub-tab 69
  - customizing a project with Citect.ini 159
  - customizing tag names 69, 72
- D**
- data concentrator
    - setup 66
  - data points
    - defining individually in an address 43
  - data replication parameter 267
  - datasets
    - adding/editing 62
  - DBF Add-In toolbar will not install 291, 304
  - deadbands 164
  - debugging
    - INI parameters to use 286, 299
  - define
    - device types and tags 41
    - enumeration 84
  - demand current summary 206
  - demand voltage summary 206
  - demo mode
    - using 219
  - design-time environment
    - described 109
  - device
    - adding to the PowerSCADA Expert project 118
    - name
      - requirements 281, 294
    - popup
      - overlapping fields 289, 302
    - profile
      - adding 70
      - creating 57
      - deleting 60
      - editing 60
      - removing from a PowerSCADA Expert project 127
    - tags
      - defining 41
    - type
      - adding, editing, deleting 44
      - defining 41
      - deleting 46
      - editing 45
      - locking 35
      - unlocking 36
  - device communications issues
    - troubleshooting 281, 294
  - device profile
    - add as default 71

- device type
    - category/subcategory
      - adding 48
      - deleting 48
      - editing 48
    - composite 65
    - drivers
      - INI parameters 162
    - drivers and protocols
      - list of supported 18
    - name
      - limitations 44
  - Device Wizard
    - before you use it 117
  - devices
    - add via CSV file 129
  - devices in the PowerSCADA Expert project
    - editing 132
  - diagnostics (Application Configuration Utility) 221
  - Diagnostics page 225
  - disabling "tag not exist" prompt. 112
  - disk storage calculations 164, 179
  - display client
    - basic report viewing 188
  - display projects button
    - adding a project in the Profile Editor 71
  - DNP3-TCP device
    - adding 123
  - DNP3 protocol support 68
  - DNP3\_serial
    - adding to PowerSCADA Expert project 120
  - documentation in this release 26
  - driver parameters
    - editing 19
    - selecting for a profile 66
    - setting up in the Profile Editor 60
  - drivers
    - choosing for device types 45
    - general PowerSCADA Expert 249
    - generic power device 18
    - how they work 18
    - subscription expirations 19
    - subscription types 18
  - duplicate devices
    - defined 19
- E**
- EcoStruxure Web Services (EWS)
    - configuration 222
      - avoid timeouts 222
    - introduction 186
  - email
    - configure to send basic reports 191
    - sending reports 192
  - encryption
    - aes file for saving INI file to a different computer 224
    - password 135, 138
  - energy readings 202
  - energy summary 202
  - engineering template
    - creating 37

- engineering units
    - adding 40
      - exceptions 39
    - adding from the Project tab 71
    - deleting 41
    - editing 41
    - multipliers and offsets 39-40
  - English.dbf
    - used in translation 120, 122, 124, 127
  - ENMPort
    - ini parameter 270
  - ENMServer
    - ini parameter 270
  - entering copy for translation 24, 119, 122, 124, 126
  - enumeration
    - defining 84
    - setting up alarm based on 155
    - status enumeration logic code 91
  - equipment column
    - alarms/events pages 172
  - equipment name
    - adding via I/O Device Manager 129
    - using when creating a one-line 112
  - equipment pop-up page 180
  - Equipment.Profiles
    - file storage location 74
    - viewing 70
  - errors
    - ATS 149
    - breaker 150
    - busbar 149
    - meter 151
    - sources 152
    - switch 149-150
    - transformer 153
  - errors and warnings
    - correcting in the graphics page 112
    - one-line 145
  - event filters 176
  - event log 173
    - acknowledge, silence, print 174
    - logging data to disk 174
    - problem with alarms not displaying 156
  - event notification service
    - Application Configuration Utility 222
  - events
    - historical logging 156
    - storage limitations 156
  - Example Automation.xls
    - using to export multiple devices 129
  - Excel DBFAdd-In toolbar will not install 291, 304
  - export
    - PowerSCADA Export 73
    - Profile Editor (backup) 74
    - project files from Profile Editor 72
    - SCL files 75
- F**
- FAQs 279, 293
    - #COM error for circuit breaker status 290, 303
    - adding/removing devices in I/O Device Manager 280, 293

- auditing user logins/logouts in the Event Log 290, 303
- browser navigation buttons do not work 285, 298
- can I change %CLUSTER% name 289, 302
- changing default pickup/dropout text 285, 298
- creating and configuring busbars 285, 298
- creating device profiles without PowerLogic drivers 280, 293
- creating users in Runtime 282, 295
- device name requirements 281, 294
- device type vs device profile 280, 293
- device writing to its registers 289, 302
- Excel DBF Add-In toolbar will not install 291, 304
- How do I troubleshoot device communications issues? 281, 294
- how to use MODBUS communications methods 281, 295
- if tag is defined, is it polled 288, 301
- INI parameters used for debugging 286, 299
- managing categories and subcategories 280, 293
- managing projects in PowerSCADA Export 282, 295
- naming conventions for servers/clusters 282, 295
- overlapping fields in device popup 289, 302
- performance tuning parameters 287, 300
- preventing reads/writes to invalid areas of memory 290, 303
- synchronize new PC with master PC 280, 293
- what are genies 283, 296
- what can be set up in logging and archiving 285, 298
- what can I modify during runtime? 285, 298
- working with templates 284, 297
- FileFormat parameter
  - historical logging of alarms and events 157
- filter
  - import data 79-80
- filter genie states 145, 175
- filter the alarm and event logs 176
- filters
  - real time and alarm 51
- firewall ports
  - opening for runtime 169
- First dbf record
  - error 291, 304
- force edit 81
- format codes
  - alarm definitions 86
  - control definitions 86
  - custom control 87
  - definitions 85
  - predefined control 87
  - real-time definitions 85
  - reset 87
- frm file
  - associated with new genie 284, 297
- FRM file
  - formatting genie appearance 115-116
- functional address 43, 71
  - definition 275
  - edit 49
- functional constraints
  - IEC 61850 77

- fundamental phasor readings 203
- G**
- G3200 setup 67
- generic I/O points
  - assigning tags 46
- generic MODBUS drivers
  - included in system 18
- genie
  - adding to a drawing 111
  - auto assign 144, 147
  - auto configure 144, 147
  - busbar 140
  - configuration
    - viewing 144
  - creating and editing 115
  - grouping 145
  - icon 111
  - in Graphics page 283, 296
  - libraries
    - ITEM1 247
    - PLS\_ALARM 239
    - PLS\_ANSI\_BASE\_1 / PLS\_ANSI\_BASE\_2 239
    - PLS\_ANSI\_CB\_1 / PLS\_ANSI\_CB\_2 241
    - PLS\_ANSI\_SW\_1 / PLS\_ANSI\_SW\_2 242
    - PLS\_DISPLAY 242
    - PLS\_GEN\_BASE\_1 / PLS\_GEN\_BASE\_2 243
    - PLS\_GEN\_CMD\_1 / PLS\_GEN\_CMD\_2 244
    - PLS\_GEN\_EQ\_1 / PLS\_GEN\_EQ\_2 244
    - PLS\_IEC\_BASE\_1 / PLS\_IEC\_BASE\_2 245
    - PLS\_IEC\_CB\_1 / PLS\_IEC\_CB\_2 245
    - PLS\_IEC\_SW\_1 / PLS\_IEC\_SW\_2 246
    - PLS\_METER 246
  - meter 141
  - repairing 147
  - resizing 141
  - source 142
  - state filters 145
  - switch/circuit breaker 141
  - transformer 142
- genie color
  - changing in project pages 111
- GenieDefinitions.xml file
  - one-line 143
- genies
  - PowerSCADA Expert 139
- GetProviderResult
  - PLSProviders.ci 160
- GetProviderStatus
  - PLSProviders.ci 160
- getting more information 26
- graphics library parameters 268
- graphics page
  - adding in PowerSCADA Export 110
  - setting as startup page 111
- Graphics runtime environment
  - editing appearance 114
- group by type or page 145
- guest user
  - single sign on 235

- 
- H**
- harmonic apparent power flows 204
  - harmonic reactive power flows 205
  - harmonic real power flows 206
  - help files 26
  - historical parameters of alarms
    - setup 156
  - historical trends
    - viewing 179
    - viewing from a one-line 182
  - HostName
    - ini parameter 269
- I**
- I/O Device Manager
    - adding a device 117
    - adding a device to a project 118
    - before you use it 117
    - editing device information in a PowerSCADA Expert project 132
    - removing device 127
  - I/O Diagnostics 225
  - ICD file
    - editing 75
  - icons
    - locked and custom 35
  - IEC
    - import filter for ICD files 77
  - IEC 61850
    - adding device to PLS project 125
    - advanced control setup 116
  - datasets
    - adding/editing 62
    - functional constraints 77
    - general workflow 25
    - perform advanced control 185
    - re-match 80
    - report control blocks 77
      - adding/editing 63
    - tag construction 83
    - tags
      - logic codes described 87
  - IED
    - defined 275
  - IF logic
    - Live View 213
  - IFE parameter 266
  - IFM parameter 266
  - ignored devices and topics 164
  - IID
    - defined 275
  - import
    - files into Profile Editor 76
    - filter data 79
    - ignore 81
    - match option 81
    - project files into Profile Editor 72
    - removing import templates 82
    - SCL files 77
    - set to unmatched 81
    - template
      - applying 76
-

- using 81
- import filter
  - IEC 61850 files 77
- import Import reconciliation screen 78
- incremental reactive energy summary 204
- incremental real energy summary 204
- INI parameters 248
  - for historical reports 188
  - for supported device type drivers 162
  - to use in debugging 286, 299
- instant trend
  - pens stop updating 180
  - viewing 180
- interlock check
  - IEC 61850 devices 185
- internationalization
  - real-time tables 216
- Internet Explorer 9 emulation
  - reporting 188
- intervals
  - trend tags 66
- invalid or missing tag address
  - correction 42
- Is Device Tag
  - reporting for composite devices 59
- J**
- JBus
  - adding 18, 45
- L**
- launching the profile editor 35
- LiveView
  - basic readings summary 201
  - conditional formulas 213
  - create menu item 210
  - create new template 211
  - data format 212
  - delete 216
  - demand current summary 206
  - demand voltage summary 206
  - does not launch (Matrikon Explorer) 201
  - duplicate 215
  - edit predefined template 210
  - energy readings 202
  - energy summary 202
  - formatting 211
  - formulas 213
  - fundamental phasor readings 203
  - harmonic apparent power flows 204
  - harmonic reactive power flows 205
  - harmonic real power flows 206
  - incremental reactive energy summary 204
  - incremental real energy summary 204
  - internationalization 216
  - introduction 200
  - lock setup 209
  - minimum/maximum readings 214
  - modify 215
  - placeholders 208, 212
  - power flow summary 202
  - save a template 210
  - save a view 208



- setup 209
  - tables list 201
  - templates 207
  - THD current summary 203
  - THD voltage summary 203
  - thresholds 214
  - update device list 208
  - update intervals 208
  - update list 208
  - update now 209
  - uptime summary 204
  - URL link to launch 207
  - view area 210
  - viewer window 207
  - views 208
  - Where's my device? 209
  - LiveView Template Editor 200
  - locked device type
    - cannot delete 46
    - unlocking 36
  - locked icon 35
  - locking a device type 35
  - lockout/tagout 171
    - enable 113
  - logging level 221
  - logic codes
    - definitions 87
- M**
- main menu options 36
  - manuals 26
  - Matrikon
    - LiveView and reporting do not launch 187, 201
  - memory device
    - one-lines 136
  - menu
    - main options 36
  - Menu Configuration 113-115
    - used to edit Graphics page menu information 114
  - metadatafields.xml
    - use to edit alarm filters 177
  - meter
    - configuring 141
    - errors 151
    - information 151
  - Micrologic
    - genie in Graphics Builder 246
    - MicrologicV INI settings 267
  - MicrologicType
    - parameter 266
  - migration utility 27
  - MODBUS communications methods 281, 295
  - multi-device reports 197
  - multiple monitor support 219
- N**
- naming ports 24
  - navigation arrows
    - runtime environment 170
  - NetworkTagsDev I/O device
    - adding 22

normally closed

control tags 43

## O

onboard alarm tag

adding 157

onboard alarms 155

one-line

adding to a graphics page 111

coloring 137

communication errors 146

PowerSCADA Expert

described 135

repairing 147

one-line animation file locations 136

One-Line Configuration Utility 144

one-line device 136

one-line engine

configuration 224

one-line flowchart 134

one-line memory device 136

one-time password 226

log in 234

opening firewall ports for runtime 169

optional parameters

CSV 131

OriginalName

CSV file 132

override

tag name 59, 65

override tag status 183

## P

pack

using to delete data 117

Pagemenu.dbf

editing 114

parameter

Septom device driver configuration settings 265

parameters

advanced tag block capabilities 259

alarm and event state 268

AlarmFormat 177, 179

AlarmSummaryLength 156

AlarmUseConfigLimits 155

Bandwidth Allocation (port level) 256

BandwidthAllocation (device level) 257

cacheRefreshTime 252

CCM (Micrologic) 266

Citect INI 248

ClientAutoLoginMode

used to set up Windows integrated user 238

CommandsBandwidth 256

data replication 267

database described 248

driver

editing 19

selecting for a profile 66

enableScatteredReads 257

EventBandwidth 256

EventIdle 265

EventTable 265

- general PowerSCADA Expert drivers 249
- graphics library 268
- HighScanRate 259
- IFE (Micrologic) 266
- IFE/IFM (Micrologic) 266
- IFM (Micrologic) 266
- initCacheTimeout 252
- initUnitCheckTime 252
- kernelStatisticUpdateRate 249
- LowScanRate 259
- maxBlockSize 258
- Micrologic 266
- MicrologicType 266
- MicrologicV INI settings 267
- minBlockSize 258
- ModbusBase 254
- module-specific packet blocking optimisation settings 266
- packet blocking optimization 257
- percentBlockFill 257
- performance tuning 256, 287, 300
- RegMode 254
- retry 250
- retryexception 250
- retrytimeout 250
- security 256
- Sepam Event Reading 265
- standbyCheckTime 251
- standbyRefreshRate 251
- statusRegister 253
- statusregistertype 254
- statusUnitCheckTime 251
- Tag Scan Rate 259
- time synchronization 163
- time zone settings 162
- TimeBandwidth 256
- timeout 250
- TimeSync 253
- timeZone 255
- usewritemultiregistersonly 249
- watchtime 249
- waveform 263
- WaveformBandwidth 256
- WaveformCheckTime 264
- WaveformMatchMargin 264
- WaveformsDisable 264
- WaveformZone 264
- Parameters file
  - changing 106
- password
  - one-time 226
- password encryption 135, 138
- pc-based alarms 154
  - setting up 59
- performance tuning parameters 256, 287, 300
- pickup/dropout text
  - changing for alarms 285, 298
- platform refresh 161
  - LiveView 209
  - reporting 196
- PLS export 73
- PLS\_ClearCache 161

- PLS\_Example project 109
- PLS\_PLatformRefresh 162
- PLS\_ReportDsp()
  - Cicode function
    - used in report configuration 187
- PLS\_ReportPage
  - used in report definition 187
- PLSAnimBusbar()
  - Cicode function for custom genie animation 116
- PLSProviders.ci module 159
- PM5000
  - device profiles for the I/O Device Manager 119, 121
- PM5000S 119, 121
- PM5000S1 119, 122
- PMCU
  - adding onboard alarms and waveform captures 156
  - defined 276
- polling rules for tags 288, 301
- port renaming
  - serial 120
- ports
  - naming 24
  - renaming
    - serial 122
    - TCP 120, 125
- power flow summary 202
- PowerSCADA Expert
  - backup 158
  - described 17
  - naming conventions 282, 295
  - project
    - creating 109
    - uninstall and reinstall 25
- PowerSCADA Expert one-lines
  - described 135
  - error logging 146
  - genie introduction 139
  - GenieDefinitions.xml file 143
  - introduction 144
  - SupportedGenies.xml file 143
- predefined control format codes 87
- predefined project
  - PLS\_Example 109
- printing
  - CSV file 47
- printing the alarm log 174
- privileges
  - user
    - PowerSCADA Expert 236
- PrivLevel
  - ini parameter 269
- profile editor
  - export 74
  - launching 35
  - overview 31
- Profile Editor
  - importing files into 76
- Profile Editor.exe.config
  - using to change trend tag scan intervals 164

profiles

- adding, editing, deleting 59
- enabling waveforms 57
- file storage location 74
- name limitations 59
- viewing 57

project

- adding 70
- cannot add 291, 304
- creating in PowerSCADA Expert database 73
- deleting 72
- editing 71
- exporting (to PowerSCADA Studio from Profile Editor) 73
- managing in PowerSCADA Expert 282, 295
- name limitations 71
- problems when renaming in PowerSCADA Studio 109
- reusing 109
- setting up 68

project parameters

- sub-tab 69
  - using to add information for imports 69

Project Parameters sub-tab 72

project setup

- introduction 19, 99

project XML files

- storage location 74

PSEHostName

- ini parameter for SSO 269

PSEHostPort

- ini parameter 270

**Q**

Quantum PLC

- genie in Graphics Builder 247
- time-stamped events 271

**R**

rack position

- displaying incorrectly 141

Rapid Access Label

- configure the report 189
- create and use 217
- label stock 218
- view LiveView tables 207

re-match IEC 61850 items 80

read logic codes 87

read/write logic codes 97

real-time format codes definitions 85

real-time lists

- viewing from genies 183

redundant I/O servers

- cluster setup 22

refreshing tags in the Project tab 70, 73

repair

- one lines 147
- project 145
- upgrade project 145

report configuration

- saving 189

report control blocks

- IEC 61850 77

reporting basic

- application tasks 188

- ReportMailer utility
    - sending email reports 192
  - reports
    - adding to runtime menu 187
    - basic
      - accessing, creating, modifying, removing 189
      - configure email to send 191
      - editing appearance 195
      - export 195
      - exporting, printing, saving 194
      - localization 200
      - set up a display client for report viewing 188
      - set up delivery method and email address 225
      - URL routing 199
    - do not launch (Matrikon Explorer) 187
    - enable Windows Authentication 198
    - feature described 187
    - INI parameters 188
    - introduction 195
    - multi device usage 197
    - PLS\_ReportDsp() 187
    - PLS\_ReportPage 187
    - sending via email 192
    - setting up in the runtime menu 187
    - single device usage 196
    - SMTP server and from address 191
    - tabular 197
    - trend 198
    - using ReportMailer 192
  - reserved names
    - do not use in projects 277
  - reset format codes 87
  - resets
    - adding custom 54
    - how they operate 43, 55
  - restart services 220
  - restore
    - project file 158
  - resx files
    - localization 200
  - reusing a project file 74, 109
  - roles
    - adding for access to LiveView 217
    - adding user 237
    - link to users in Project Setup 102
    - link users in Project Setup 102
  - RPC
    - set to TRUE for YubiKey 228, 235
  - runtime environment
    - described 18
    - launching and closing 169
    - viewing 170
- S**
- SBOw tag
    - creating 291, 304
  - SCADA
    - defined 16
  - scan intervals
    - trend tags 163

- scattered reads
  - using enableScatteredReads parameter 257
- SCL
  - defined 278
  - export 75
- SCL files
  - importing 77
- screen resolution
  - Profile Editor 36
- security parameters 256
- security viewer 178
  - filter 178
- select before operate
  - setup 291, 304
- Select Devices
  - LiveView 208
- Selected Device Profiles
  - sub-tab 69
- selecting 39
- Sepam
  - genie in Graphics Builder 247
- Sepam device driver configuration settings 265
- sequence of events
  - alarm/event logs 176
- SER (sequence of events recorder) 18, 65, 176-177
- serial device
  - adding to PowerSCADA Expert project 120
- servers/clusters
  - naming conventions 282, 295
- service inventory 221
- setpoints and delays
  - entering 155
  - entering from the runtime environment 182
- setting up
  - alarm based on an enumeration 155
  - audible alarms 157
  - device type categories 47
  - normally closed/normally open controls 43
- severity
  - changing for alarms 156
- show by type or page 145
- sim source
  - transformer top and bottom 153
- single device reports 196
- single sign on 235
  - guest user 235
- SMTP server
  - for reports 191
- sorting the alarm log 58, 172-173
- source
  - configuring 142
  - errors 152
  - information 151
- start and stop AdvOneLine 139
- startup page
  - setting 111
- state filters 145
- storage limitations
  - calculating disk space 164, 179
- subscriptions
  - drivers 18

- support
    - technical 26
  - SupportedGenies.xml file
    - PowerSCADA Expert one-lines 143
  - switch
    - configuring 141
    - errors 150
    - information 148, 150
  - symbol
    - creating custom for custom genie 116
  - synchro check
    - IEC 61850 devices 185
    - set up
      - IEC 61850 116
  - synchronize new PC with master PC 280, 293
- T**
- tabular usage reports 197
  - tag
    - custom
      - adding 50
      - customizing the name 69
      - IEC 61850 described 83
      - refreshing in the Project tab 70
  - tag-polling rules 288, 301
  - tag addresses
    - adding 56
    - control 55
    - correcting missing or invalid 42
    - editing 52, 56
      - generic 56
    - onboard alarm 53
    - real-time 52
    - reset 54
  - tag count
    - driver subscriptions 18
  - tag definitions
    - printing the CSV file 47
  - tag names
    - customizing 72
  - tag override 59, 65
  - tag status
    - overriding 183
  - tag types 83
  - tag viewer
    - setup 158
    - viewing 184
  - tags
    - assigning to generic I/O points 46
    - scanning intervals for trend tags 66
  - TCP device
    - adding to PowerSCADA Expert project 120, 125, 127
    - adding to project 118
  - technical support 26
  - Template Editor
    - LiveView 200
    - lock 209
  - templates
    - creating 37
    - deleting 39
    - imports 81
    - working with 284, 297



- THD current summary 203
  - THD voltage summary 203
  - time-stamped events
    - Quantum PLC 271
  - time quality
    - alarm/event logs 176
  - Time Sync
    - filter for alarm or event log 177
  - time synchronization parameter 163
  - time zone setting parameter 162
  - transfer switch
    - configuring 140
  - transformer
    - configuring 142
    - errors 153
    - information 152
    - top and bottom 153
    - voltage-level and pass-through 137, 143, 153
  - translation
    - using description fields 24
  - trend tag intervals 179
    - changing in the Profile Editor 61
  - trend tag scan intervals 66, 163
  - trend usage reports 198
  - trending (Analysis Page)
    - described 179-180
  - trends
    - maximum number of pens 179
    - setting up in Profile Editor 59, 61
    - viewing from within a one-line 182
  - two-factor authentication 226
  - typical workflow
    - creating a graphics page 108
    - creating a project in Profile Editor 69
    - Profile Editor 31
- U**
- uninstalling and reinstalling PowerSCADA Expert 25
  - unique ID
    - determining for onboard alarms 54, 157
  - units (engineering) and conversions
    - creating or editing a units template 37, 39
    - deleting 41
    - deleting a unit template 39
    - editing 41
    - exceptions 39
  - uptime summary 204
  - URCB
    - entering for IEC 61850 device 127
  - URL
    - visit to send reports via email 192
  - URL link
    - launch Live View Viewer 207
  - URL reporting for basic reports 199
  - UseHTTPS
    - ini parameter 270
  - UsePLSFilter
    - alarm/event filter parameter 176, 255
  - user access
    - adding and modifying 236
    - changing rights 238
    - Windows integrated 237

- useraccess
    - default settings 236
  - users
    - add in Project Setup 102
    - creating in Runtime environment 282, 295
  - using block writes 98
- V**
- value type
    - controlling real-time display 181
  - Variable tags not found
    - disabling the prompt 112
- W**
- waveform
    - why can't lacquire? 291, 304
  - waveform parameters 263
  - waveforms
    - enabling in Create Device Profiles tab 57, 70
    - enabling onboard 156
    - linking to alarms 166
    - management 166
    - special digital tags 167
    - storage rules 166
    - viewing from the Alarm Log 174
    - viewing from the runtime environment 173
    - viewing in the runtime environment 182
  - WaveWeb waveform viewer 174
    - PDF location 182
  - web.config
    - securing a URL for sending reports via email 192
    - SMTP server used when sending email 191
    - specify Windows Authentication, adding roles for LiveView 217
    - specify Windows Authentication, adding roles for Reporting 199
    - specify Windows Authentication, adding roles for Live View 217
  - WebReachServer
    - ini parameter 269
  - Where's my device? 209
  - Windows Authentication
    - enable for Live View 216
    - enable for reports 198
    - in Project Setup 103
  - Windows integrated user accounts 237
  - Windows security
    - using to remove user access rights 236
  - write logic codes 97
  - writes (logic codes) 97
- X**
- XML files
    - storage for project files 74
- Y**
- YubiKey 228
    - add the Citect parameter 228
    - disable 234
    - import and import password settings 233
    - programming 230
- Z**
- zOL
    - memory device for one-lines 136



**Schneider Electric**  
35, rue Joseph Monier  
CS 30323  
F - 92506 Rueil Malmaison Cedex  
[www.schneider-electric.com](http://www.schneider-electric.com)

PowerLogic is a trademark of Schneider Electric in France, the USA, and other countries. Other trademarks are the property of their respective owners.

This product must be installed, connected, and used in compliance with prevailing standards and/or installation regulations. As standards, specifications, and designs change from time to time, always ask for confirmation of the information given in this publication.

7EN02-0403-00 03/2017

Replaces 63220-100-200J1, dated 03/2016

© 2015 Schneider Electric. All Rights Reserved.