SmartPlant Electrical
User's Guide

Process, Power & Marine

INTERGRAPH

Table of Contents

Preface

This user's guide describes concepts, procedures, and interface features of SmartPlant Electrical.

Send documentation comments or suggestions to PPMdoc@intergraph.com.

Using SmartPlant Electrical: An Overview

SmartPlant Electrical enables electrical engineers, designers, maintenance personnel in EPCs, and Plant Owners to design and maintain the electrical equipment and documentation for process plants, refineries, and so forth. SmartPlant Electrical is designed to meet specific needs of electrical engineers by automating work processes, complying with corporate or project standards and manipulating electrical data throughout the plant and project lifecycle.

The software allows you to create electrical equipment and components and then dynamically associate among them. You can determine the load requirements in your plant, keep track of your electrical inventory, and establish the electrical relationships among the different components.

You can use SmartPlant Electrical to generate single line diagrams, schematic drawings, wiring drawings, and cable block diagrams without an external CAD engine and thus generate these deliverables automatically, creating graphical reports based on the engineering work itself. Electrical engineering deliverables include electrical equipment list reports, consumer list and load summaries, power distribution board design, single-line diagrams, cable block diagrams, and schematic drawings, as well as cable lists and wiring reports.

SmartPlant Electrical supports the major commercial databases and outputs drawings in common CAD formats including SmartSketch, and generates reports in Microsoft Excel format.

SmartPlant Electrical incorporates the latest Microsoft technologies, such as OLE automation, to provide integration with existing data and other systems. Running on Microsoft Windows NT and Windows 2000, SmartPlant Electrical does not require a traditional, expensive CAD engine for the creation of single line diagrams and schematics. The open architecture of SmartPlant Electrical permits integration with other systems, such as Intergraph SmartPlant Instrumentation, EDSA, and ETAP, all of which allow users to share data with third-party software.

In the SmartPlant Electrical main window, the following windows are available:

Electrical Index

Enables you to quickly create, modify, view, and select items in your project inventory. For more information, see *Working with Electrical Index: An Overview*, page 46.

Reference Data Explorer

Enables you to create your default electrical equipment and data, loads, panels, and cables, which you can then use to create and populate the project database in an efficient and consistent manner. The **Reference Data Explorer** enables you to define equipment profiles, a powerful means of creating standard data. You can apply this data in batch mode to a large number of items of the same type, such as motors, resulting in the creation of project items and populating them with default data. For more information, see *Working with Reference Data Explorer: An Overview*, page 44.

Electrical Engineer

Enables you to create, modify, and display the electrical relations between various electrical items. In this way, you establish the electrical infrastructure. For more information, see *Working with Electrical Engineer: An Overview*, page 78.

Properties Window

Displays the data fields of the currently selected item in tabular style where you can edit the property values as needed. For more information, see *Working with the Properties Window: An Overview*, page 108.

Drawing View

Provides a graphical presentation of templates, power distribution board layouts, single line diagrams, or schematics.

Tabular Editor

Enables viewing and editing of items of the same type, and batch assignment of properties to a selected group of items. You can also use the Tabular Editor to select items for including in reports. For more information, see *Working with the Tabular Editor: An Overview*, page 96.

Status Bar

Displays summary information about selected menu items, processes, and tips when performing particular actions. For more detailed information, refer to the appropriate SmartPlant Electrical Online Help topic.

New Features in SmartPlant Electrical Version 2007

General Features

Creating Sub-Folders in the Electrical Index and Reference Data Explorer

SmartPlant Electrical enables you to add sub-folders to the folders in the **Electrical Index** and **Reference Data Explorer**. This makes it easier to find items in a folder that holds a long list of items.

Deletion Preferences

When deleting an item, the software allows you to automatically delete various items that are associated with the item you want to delete. You can set a preference that instructs the software to delete associated items that belong to the item types that you select to delete.

Showing Related Items in a New Window

SmartPlant Electrical allows you to select an item in the **Electrical Index** or **Electrical Engineer** and show all the items that are related to the selected item in a new window. For example, if you select a motor, the software displays the associated control station and feeder circuits grouped together in a new window. This way you do not need to search for associated items and save time when wiring your project items.

The Description Property Length

The length of the **Description** property in SmartPlant Electrical has been increased to 4000 characters.

Deleting Loads Together With Their Associated Items

A new preference now allows you to delete a load together with its associated child items, for example feeding power cable, schematic drawing, and so on.

Cables

Cable Block Diagrams

You can create a graphical environment that displays the current interconnectivity. You can create new interconnectivity, between cables and electrical equipment in your plant. Using drag and drop operations, position items, connect existing cables, or add new cables and electrical equipment to your drawing.

Wiring Project Items

SmartPlant Electrical enables you to perform wiring and create wiring drawings. The software provides for the creation of basic wiring and enables you to make the terminations that you require. You can do the following:

- Create terminal strips for all types of applicable equipment.

- Create terminals on terminal strips.

- Manage both cable sides.

- Manage both conductor sides.

- Create wiring drawings and make connections.

- Generate termination reports.

Inclusion of Voltage Drop at Starting Current for All Power Cables

When sizing power cables, it is now possible to enter starting current for all power cables and not just for the ones that are connected to motors.

Connecting Control, Instrumentation, and Grounding Cables to Both Circuit Sides

SmartPlant Electrical no longer restricts the sides of control, instrumentation, and grounding cables to connect only to certain item types. Unlike power cables that have a "power source" and "load" side (power flow convention) convention, there is no such practice for other cable categories. Therefore, all item types are now available as possible and valid equipment at both sides of cables. Hence, you can now interconnect any type of equipment.

Automatic Definition of Cable Splices

The software allows you to automate the definition of cable splices for cables that exceed the maximum cable drum capacity. When assigning drums to cables, the software now automatically defines the required splices and specifies the lengths of the cable cuts.

Ability to Enter an FLA Code Factor for Cable Sizing

When entering sizing data for a cable and selecting **Yes** from the **Use code factor** list, you can now enter a **Code factor** value that the software will use as a multiplying factor that will suit the connected load.

Cable Routing

NEC Segment Fill Validation Method

SmartPlant Electrical provides a new option which allows you to perform segment fill validation using the National Electric Code (NEC) method. The validation method that was used in the previous version is now called the Conventional method.

Converting Equipment

Calculating FLA Values of a Secondary Transformer Winding

SmartPlant Electrical calculates FLA values of a secondary transformer winding exactly the same way it does for the primary winding. The software automatically makes the secondary rated power values identical to the primary.

Automatic Calculation of Secondary Rated Power for Two-Winding Transformers

For two-winding transformers, the software automatically copies the secondary rated power from the primary rated power.

Current Limiting Reactor - A New Item Type

SmartPlant Instrumentation has introduced a new item type - Current Limiting Reactor. You can create new current limiting reactors in the **Converting Equipment** folder in the **Electrical Index** and **Reference Data Explorer**.

Resistive and Reactive Power Losses

When entering electrical data for a converting equipment item on the **Common Properties** dialog box or in the **Properties** window, you can now enter values for the resistive and reactive power losses that occur due to the copper and iron losses. The software can use these values to account for converting equipment power losses when running bus load calculations for batch load assignment and All Feeder Load Summary reports. The software will use these values if you set your report preferences to account for these losses.

Documents

Associated Drawings

A new short-cut menu allows you to navigate from an item directly to any of its related drawings, without the need to move to the document folder.

Electrical Engineer

Showing the Association of Loads That Are not Connected to Circuits

The **Electrical Engineer** now shows loads as child items of buses even if the loads are not connected to circuits.

Connecting Multiple Loads to Stand-Alone Circuit Breakers

The software now allows you to connect multiple loads to a stand-alone circuit breaker.

Electrical Index

Current Limiting Reactors — A New Item Type

SmartPlant Electrical introduces a new type of converting equipment that allows you to limit the short-circuit power of networks or installations to an acceptable value with regard to the short-circuit strength of the equipment or the breaking capacity of the circuit breaker.

Using Coupler Riser Circuits to Interconnect Buses in Different PDBs

SmartPlant Electrical now enables you to use coupler riser circuits to interconnect buses that belong to different power distribution boards.

Creating Stand-Alone Buses

SmartPlant Electrical now allows you to create stand-alone buses that are not associated with power distribution boards. You create these buses in the **Substations** folder of the **Electrical Index**. Stand-alone buses can be parent items of circuits and circuit internals, but not cells.

Two New Cell Properties

Cells now have two new properties that define the cell location in the PDB layout: **Section Name** and **Row Name**. You can define these properties in the Properties window for cells.

Interface to SmartPlant Instrumentation

Associating Electrical Tags Created in SmartPlant Instrumentation

You can publish electrical equipment without signals from SmartPlant Electrical and associate that equipment with electrical tags created in SmartPlant Instrumentation. Likewise, you can retrieve electrical signals originating in SmartPlant Instrumentation for association with electrical equipment in SmartPlant Electrical.

Load Summaries and Load Assignments

Accounting for Converting Equipment Power Losses

When running bus load calculations, the software calculates the total electrical consumption taking into account converting equipment power losses. The software adds these power losses to the upstream feeding bus and accounts for them as continuous loads. The same losses are also accounted for in the All Feeder Load Summary and PDB Load Summary reports. You can enter appropriate power losses values on the **Electrical Data** tab of the **Common Properties** dialog box for any converting equipment item.

Calculating and Storing the Results in the Database

SmartPlant Electrical now uses a new algorithm to calculate bus load flows. The software stores the calculated circuit totals in the database and enables you to generate an enhanced All Feeder Load Summary report. The calculation is based on either consumed or rated power values which are compensated or uncompensated for coincidence factors of either the relevant buses or the individual loads.

Moving Loads with the Feeding Circuits to Other Buses

When moving a load to another bus, the software now enables you to move the load together with its feeding circuit and the circuit internal items. This is true for all the loads apart from those that are connected to the bus through converting equipment.

Showing Calculation Preferences When Running Batch Load Assignment

You can now view the calculation preference settings when effecting batch load assignment. This can help you understand the calculated results without the need to close the **Batch Load Assignment** dialog box and open the **Preferences** dialog box.

Calculating Bus Line Current and Power Factor at Run and Peak Conditions

When running batch load assignment, the software calculates the bus line current and power factor values at run and peak conditions. The difference between the two conditions is that the peak calculation takes into account the spare and standby loads associated with the current bus.

Correcting Bus Power Factors Using Capacitor Banks

When running batch load assignment and calculating the electrical consumption for a bus, the software allows you to calculate the power factor of a capacitor bank that needs be associated with the selected bus so that you can correct the power factor of that bus. Creating a capacitor bank that has the power factor you just calculated and associating this capacitor bank with the current bus corrects the bus power factor when calculating the electrical consumption of the bus.

Batch Load Association Shows Load Operating Mode

The **Batch Load Association** dialog box now shows the operating mode of the associated loads.

Batch Load Association Allows the Display of Preferences

You can now view the settings of your calculation and report preferences when the **Batch Load Association** dialog box is open.

Lookup Tables

Associating and Applying Lookup Tables to Circuit Internal Components

The software now allows you to associate lookup tables with circuit internal components. You can make this association on the **Typical Circuit Common Properties** dialog box in the **Reference Data Explorer** and then apply that typical circuit to a number of project circuits. Alternatively, you can make this association on the **Circuit Common Properties** dialog box for a circuit that you select in the **Electrical Index** and then run the **Apply Lookup Tables** feature on the **Apply Options** dialog box to propagate the lookup table data to the circuit internal components.

Automating the Sizing of Circuit Internal Components Based on Load Data

SmartPlant Electrical enables you to create lookup tables that include load properties as key fields. You can associate these lookup tables with circuits or circuit internal components and apply these lookup tables to project circuits or circuit internals to populate the lookup table data. This feature is especially useful when creating protection equipment, fuses, circuit breakers, and so forth whose properties correlate with load lookup tables.

Options Manager

Delete Loads

You can now set which related (sub) items of a main item are deleted when deleting a main item in SmartPlant Electrical.

Cable Fill Follows NEC Regulations

Allows the use of NEC regulations in SmartPlant Electrical, when calculating and validating cable fill in cableways.

Define Item Type Symbology

You can now define the style, color and other properties of a symbol and associate it with an item type to be displayed in SmartPlant Electrical.

Reference Data Explorer

Applying Additional Regional Standard Lookup Tables

You can now apply additional regional standard lookup tables to your equipment items. In the previous version of SmartPlant Electrical, only NEC standards were available. Now, in the Data Dictionary, you can define IEC and CEC standards and then create lookup tables in the **Reference Data Explorer** based on these regional standards. You can then apply the new lookup tables to the appropriate equipment items that conform with these regional settings.

Reports

Enhanced All Feeder Load Summary Report

You can now generate a new all feeder load summary report based on the values that the software calculates and stores in the database. The report provides a load summary of rolled up buses and loads showing consumed or rated power values, compensated or uncompensated for coincidence factors of the buses or individual loads. The enhanced report displays the kV and kVAR values for individual loads grouped according to the relevant operation mode of the load. Also, the report includes the converting equipment power losses values and circuit totals.

Showing Plant, Client, and Contactor Information in Reports

When generating a report in SmartPlant Electrical, you can now show the plant and plant group information. Also, you can include the contractor name and number.

New Shipped Reports

The following new reports come shipped with SmartPlant Electrical 2007:

Cable Schedule - with issue — Lists all the existing project cables for publishing to the integrated environment.

Instruments and Cabinet Load List - with issue — A report that is intended for publishing instruments and cabinets that have been retrieved from SmartPlant Instrumentation as loads to the integrated environment.

I/O Signal and Equipment - with issue — A report that is intended for publishing signal and equipment data, or just equipment data to the integrated environment.

I/O Signal List - with issue — A report that is intended for publishing signal data to the integrated environment.

Customizing the All Feeder Load Summary Reports

You can now customize your shipped All Feeder Load Summary reports by adding or removing columns.

Rule Manager

Enhanced Copy Features

SmartPlant Electrical Rule Manager has been enhanced to provide the ability to copy property values across items, resulting in the propagation of data from one item to its related/connected item.

Propagating Data Upstream or Downstream to Related Items

Now Rule Manager has the ability to propagate properties of items from one to the other, for example load data up to the feeder circuits internals.

Propagating Load Data into Circuit Internals

Rule Manager now supports the creation of rules that can validate connected equipment to the circuit internals and whether there is an inconsistency between the size of the load and its protection device.

Schematics

Matching the Typical Schematics of Circuit to the Connected Equipment

Connected equipment and the circuit to which they are connected must have the same typical schematics. SmartPlant Electrical Rule Manager now compares the typical schematics of the two item types and informs you of any inconsistencies between the two.

Single Line Diagrams

New Placement Method

SmartPlant Electrical now provides a drag-and-drop interface for the placement and positioning of buses, generators, transformers, and other main electrical equipment on a drawing. Once the items have been positioned on a drawing the software will automatically make the connections between the items.

Include Loads

Single line diagrams now come with the option to show the connected loads and any space heaters associated with motors. The space heaters will be properly displayed, adjacent to the motors and connected to the busses that feed the heaters.

Current and Potential Transformers

SmartPlant Electrical now has the ability to create current transformers associated with the neutral of the transformer windings, distinguish from the ones associated at it's inlets/outlets and properly present these items and relations in schematics and SLDs.

Automating Meter and Relay presentation

SmartPlant Electrical supported the ability to insert dumb graphical symbols of metering equipment into an SLD and to connect between them and other objects by connecting predefined connection points. Up until now those inserted symbols and their relationships were not stored in the project database and would be lost if the drawing was deleted. Now you can show, properly position, and define the logical relations between metering and relaying equipment and any other piece of equipment in the SLD, and then save it to the project database.

Space Heater Symbols

Space heaters now have their own dedicated symbol to properly present them as an individual object and not part of a motor. Also a dedicated space heater filter is supplied with the software to distinguish it from regular stand-alone heaters.

Symbol Assemblies

SmartPlant Electrical supports the display of symbol assemblies in a single line diagram. These assemblies are used to illustrate complex items such as circuits with their internal components, or motors and their associated items.

Show the Internal Components of a Bus Tie in a Horizontally Orientated Bus Tie

A unique symbol name has been created for the internal components of bus ties that are to be shown in a horizontal orientation. The software searches for this unique symbol name and then displays the horizontal bus tie correctly on your SLD.

SmartPlant Integration

New Schema Mapping Options

It is now possible to define schema mapping for new values in enumerated lists (select lists) as well as for other properties.

Retrieving Additional Items from SmartPlant P&ID

Items that are published for SmartPlant P&ID, other than motors and heaters, are retrieved in SmartPlant Electrical under **Other Electrical Equipment**.

Publishing of All Reports

It is now possible to register any shipped or custom report for publishing.

Upgrade Schema

A new command, **Upgrade Schema**, is available on the **SmartPlant** menu. This command updates the SPELMap.xml file with schema changes that you may have made in a previous version of the software.

Symbology

Associating Symbols

SmartPlant Electrical now supports the association of a symbol with a single specific item, with item types, and with sets of items in single line diagrams, cable block diagrams, and wiring diagrams. Using the supplied system symbols, or customized symbols created by you, it is now possible to associate different symbols to different items and drawing types.

Connecting to a Plant Group: An Overview

Before you can work with SmartPlant Electrical, you need to connect to a plant group. A plant group is a specific group available within a plant hierarchy, for example, Plant1, Area1, Unit1. Plant, area and unit are examples of plant group types. The available plant group types depend on the plant hierarchy you define in Engineering Manager.

You can open a plant structure and then connect to a plant group. Also, if you have appropriate access rights, you can connect to a plant group located on a different site server. You can only connect to one plant group at a time.

Related Topics
- *Connect to a Plant Group*, page 33
- *Open Plant Structure Dialog Box*, page 583

Connect to a Plant Group

1. Click **File** > **Open Plant Group**.

   💡 **Tip**

   - If the **Open Plant Group** command is not accessible, close all Options Manager windows.

2. Do one of the following:

   - Expand the plant hierarchy and then select the desired plant group located on the lowest hierarchy level.

   - Click **Select Plant** and then, under **Available plant structures**, select a plant to connect.

   💡 **Tip**

   - If you want to connect to a plant in a different database, in the **Open Plant Structure** dialog box, click **Site Server** and from the **Files of type** list, select the desired .ini file.

Related Topics
- *Connecting to a Plant Group: An Overview*, page 32
- *Open Plant Structure Dialog Box*, page 583

Changing Plant Group Hierarchy Names: An Overview

You change the hierarchy group names of the areas and units in your project using SmartPlant Engineering Manager. A message is displayed at the start of SmartPlant Electrical and SmartPlant Options Manager informing you that changes have been made to the hierarchy names and that you should run the **Apply Naming Conventions** command in SmartPlant Electrical to complete these changes. Using the **Apply Naming Conventions** command the changes made in the hierarchy names are propagated to the item type names through the entire project according to the definitions created in the Options Manager.

- *Apply Plant Group Hierarchy Name Changes*, page 34

Apply Plant Group Hierarchy Name Changes

1. Change the Plant Group Hierarchy names in SmartPlant Engineering Manager. For more details, see Change Plant Group Hierarchy Names in the *SmartPlant Engineer Manager User's Guide*, under *Plant Structure Root*, *Plant Group Nodes*, *Change Plant Group Hierarchy Names*.

2. Click **Tools** > **Apply Naming Conventions**.

3. At the prompt, do one of the following:

   - Click **Close** to continue to work in SmartPlant Electrical.

   - Click **View Log** to open a log of the changes made to the hierarchy.

📝 **Note**

   - A message is displayed at the start of SmartPlant Electrical informing you that changes have been made to the naming conventions. This message reappears every time SmartPlant Electrical is run until you use the **Apply Naming Conventions** command.

   - Due to the possibility that the naming convention and inter item tag dependency can be very complex the software runs the **Apply Naming Conventions** twice. After running the update there may still be items that have not been updated. You can run the **Apply Naming Conventions** again. You can also update the item names by using the **Comply with Naming Conventions** in the **Plant-Wide Parameters** dialog box in SmartPlant Options Manager.

- The changes are made to the item types in a specific order. The running order for the item type name changes is: BatteryBank - BatteryCharger - Busway - Cabinet - CableDrum - CableWay - Capacitor - CurrentLimitingReactor - Generator - HarmonicFilter - Heater - HeatTrace - JunctionBox - LocalPanel - Motor - OffSitePower - OtherConvertingEquipment - OtherElectEquipment - PowerDistributionBoard - Resistor - Transformer - UPS - VariableFrequencyDrive - Bus - CableWaySegment - ControlStation - Instrument - Ammeter - Circuit - CircuitBreaker - Contactor - CurrentTransformer - DisconnectSwitch - Fuse - Multimeter - OtherDisconnectEquipment - OverloadRelay - PotentialTransformer - ProtectionRelay - SignalRun - Starter - Voltmeter.

Related Topics

- *Changing Plant Group Hierarchy Names: An Overview*, page 33

Working with SmartPlant Electrical Common Tasks

When creating or modifying data in the **Reference Data Explorer** or the **Electrical Index**, there are a number of commonly used tasks that you perform throughout the application.

Create an Item

You can create electrical loads, cables, junction boxes, and other equipment, either by creating items on-the-fly in the **Electrical Index**, or by creating typical items in the **Reference Data Explorer** and using those typical items as templates for new items or for *profiles*.

For more information, see:

- Create a Reference Item
- Create a Project Item

Working in a Multi-User Environment

Special considerations apply when several users work on the same item. For more information, see *Working in a Multi-User Environment*, page 37.

Edit Item Properties

When any item has focus in the **Reference Data Explorer** or the **Electrical Index**, you can edit the item properties in the **Properties** window. For some of the items, you can edit the most commonly used properties more conveniently on a dialog box. For more information, see *Edit Item Properties*, page 38.

Duplicate an Item

You can duplicate an item in the **Reference Data Explorer** or the **Electrical Index**. This method is advantageous when you have items that include a hierarchy of sub-items, in which case the software duplicates the sub-items with the parent item. For more information, see *Duplicate an Item*, page 38.

Rename an Item

You can rename any item that has focus. For more information, see *Rename an Item*, page 39.

Delete an Item

You can delete items in the **Reference Data Explorer** or the **Electrical Index**. When you delete items, the software deletes them from the database. For more information, see *Delete an Item*, page 40.

Select a Set of Tags

You can make a multiple selection of tags for performing editing operations in the **Reference Data Explorer** or the **Electrical Index**. For more information, see *Select a Set of Tags*, page 41.

Create a Project Item

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder or parent item where you are going to add the new item, right-click, and then click **New**.

3. Accept the default item tag that is based on the *naming convention* for the item type, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the item properties in the **Properties** window as required.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create the item.

- Some items have a **Common Properties** dialog box from which you can edit the most common properties for the item. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the item. For full details of the properties, see the appropriate dialog box topic.

Related Topics
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38
- *Item Tag Dialog Box*, page 720
- *New Item Dialog Box*, page 582
- *Rename an Item*, page 39
- *Working with SmartPlant Electrical Common Tasks*, page 34

Working in a Multi-User Environment

The software allows several users to concurrently access and edit the same item. In this case, the software accepts the changes for the first user to commit those changes. The software then saves the data committed by the second user overwriting the previously committed data.

Also, if a user deletes an item and commits the deletion, the software rejects changes made by other users and those users cannot subsequently update the item because the deletion takes priority. If you get a message that another user has modified or deleted a item that you are working on, click **Refresh** ⟳ or select another item and then reselect the item you were editing (if it has not been deleted).

Related Topics
- *Using SmartPlant Electrical: An Overview*, page 20
- *Working with SmartPlant Electrical Common Tasks*, page 34

Edit Item Properties

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Edit the item properties as required in the **Properties** window. Note that an asterisk beside a property name indicates required data.

✎ **Notes**

- For some items, you can edit properties in either the **Properties** window or on a **Common Properties** dialog box. You can click the **Common Properties** command on the shortcut menu for an item to open a dialog box on which you can edit the most commonly used properties for the item. For full details of the properties, see the appropriate dialog box topic.

- If you edit a profile, a typical circuit, or a control station in the **Reference Data Explorer**, the software uses the latest update when you next apply options to a project item. For example, if you modify the components of a typical circuit that already has a reference from a motor, the software does not automatically update the change in the typical circuit connected to the motor. The software updates the change only when you next apply a typical circuit or a profile to the motor.

Related Topics
- *Create a Project Item*, page 53
- *Delete an Item*, page 40
- *Rename an Item*, page 39

Duplicate an Item

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Right-click an item, and on the shortcut menu, click **Duplicate**.

3. Accept the default item tag that is based on the *naming convention* for the item type, or type your own item tag according to the naming convention.

    💡 **Tip**

    - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. For the item that you duplicated, edit the item properties in the **Properties** window.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you duplicate the item.

- Some items have a **Common Properties** dialog box from which you can edit the most common properties for the item. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the item. For full details of the properties, see the appropriate dialog box topic.

- When you duplicate an item, the software copies only the item properties, except for the **Item tag** property, which follows the item naming convention. The software does not copy associated items such as cables, typical circuits, or control stations, nor does it duplicate associations to feeder circuits.

- For an item that includes sub-items, such as a circuit with internal components, the software duplicates the sub-items as well. In this case, the sub-items are the circuit components.

Related Topics
- *Apply Lookup Table Data to Equipment*, page 143
- *Create a Project Item*, page 53
- *Creating Electrical Equipment Common Tasks*, page 75
- *Delete an Item*, page 40
- *Edit Item Properties*, page 38
- *Rename an Item*, page 39
- *Working with SmartPlant Electrical Common Tasks*, page 34

Rename an Item

1. In the **Reference Data Explorer**, the **Electrical Index**, or the **Electrical Engineer**, expand the folder hierarchy by clicking the + icons or double-clicking the folders.

2. Select an item and open the **Item Tag** dialog box by doing one of the following:

   - Right-click the item, and on the shortcut menu, click **Rename**.

   - Press F2.

3. Type a new item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

   📝 **Note**

   - If you rename a profile, a typical circuit, or a control station in the **Reference Data Explorer**, the software uses the new name if there is a reference to one of these items from a project item. For example, if you change the name of a typical circuit that already has a reference from a motor, the software automatically updates the name of the typical circuit associated with the motor.

   Related Topics
   - *Create a Project Item*, page 53
   - *Delete an Item*, page 40
   - *Duplicate an Item*, page 38
   - *Edit Item Properties*, page 38
   - *Working with SmartPlant Electrical Common Tasks*, page 34

Delete an Item

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Right-click the item that you want to delete, and on the shortcut menu, click **Delete**.

3. At the prompt, click **Yes** to confirm the deletion.

📝 Notes

- You can delete only items that do not have associations to other items. For example, you cannot delete a power distribution board associated with loads unless you first dissociate the loads.

- If you select more than one item for deletion, any failures to delete items are recorded in the **Batch Deletion** log file. For the procedure for creating and viewing the log file, see *Generate Error Log Files*, page 120.

Rules for Deleting Reference Items

The following rules apply when you delete an item in the **Reference Data Explorer**:

Item	Precondition	Result
Profile	The profile is not the most recent one that you applied to any motor.	The software deletes the profile only.
Typical circuit	You have dissociated the typical circuit from all profiles to which it belonged and the typical circuit is not the most recent one that you applied to any electrical equipment.	The software deletes the typical circuit and all the circuit components.
Typical control station	You have dissociated the typical control station from all profiles to which it belonged and the typical control station does not have a link with any electrical equipment.	The software deletes the typical control station and dissociates any control station cables.
Typical schematic	You have dissociated the schematic from all profiles and project loads to which it belonged.	The software deletes the typical schematic only.
Typical block	You have dissociated the block from all schematics to which it belonged.	The software deletes the typical block only.
Typical motor	You have dissociated the motor from all profiles to which it belonged.	The software deletes the motor and any space heater associated with the motor.

Rules for Deleting Project Items

The following rules apply when you delete an item in the **Electrical Index**:

Item	Precondition	Result
Motor	None.	The software deletes any space heater associated with the motor and clears any **To** side cable association. The software also clears any relationships to control stations.
Static load	None.	The software clears any **To** side cable association. The software also clears any relationships to control stations.
Converting equipment	None.	The software moves any items in the electrical hierarchy below the deleted converting equipment to the root of the **Electrical Engineer**. The software also clears any **From** or **To** side cable association.
Panel	You have dissociated all the circuits in the panel from external items, such as cables, loads, and so forth.	The software deletes all the internal panel circuits and moves any items in the electrical hierarchy below the deleted panel to the root of the **Electrical Engineer**.
Panel circuit	None.	The software moves any items in the electrical hierarchy below the deleted panel circuit to the root of the **Electrical Engineer**.

Related Topics
- *Delete a Cable*, page 189
- *Delete a Power Distribution Board or a PDB Internal Item*, page 156

Select a Set of Tags

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Right-click a folder and adjust the size of the panes to view the items in the list view pane.

3. If required, change the view in the list view pane by clicking **View Settings** to toggle through each view of the items in the list view pane, or click the arrow beside the icon and select the desired view.

4. Select one or more items in the list view pane. Hold **Ctrl** to select several items or hold **Shift** to select a range of items.

Note

- When you select a set of data, the list at the top of the **Properties** window displays **Select set** and you can type properties that apply to all the items. Also, you can select a specific item from the list to view or edit its individual properties.

Related Topics
- *Edit Item Properties*, page 38
- *Rename an Item*, page 39
- *Working with the Tabular Editor: An Overview*, page 96

Show Related Items in a New Window

1. In the **Electrical Index** or **Electrical Engineer** right-click an item.

2. On the shortcut menu, click **Show Related Items in a New Window**.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Working with Wiring: An Overview*, page 329

Working with the Reference Data Explorer: An Overview

You use the **Reference Data Explorer** for creating reference items that you use as templates for your SmartPlant Electrical items in the **Electrical Index**.

The main features of the **Reference Data Explorer** are:

Toolbar

The toolbar contains the following options:

Find - Click **Find** to open the **Find** dialog box. Click **Close** to close the active **Reference Data Explorer** window.

Refresh - Click **Refresh** to update the **Reference Data Explorer** display. This feature is useful where multiple users are working on the same set of data.

Views - Click **Views** to toggle through each view of the items in the list view pane, or click the arrow and select the desired view.

Buttons - Buttons provide a useful shortcut to an item or a group of items. Click **Buttons** to toggle show / hide for existing buttons. To create a button for the currently selected item or folder, click the arrow and click **Add Button**. To remove a button, select the button to remove, then click the arrow, and click **Remove Button**. Note that if you do not select any button to remove, the software removes the uppermost button in the list.

Tree View Pane

The tree view pane of the **Reference Data Explorer** shows typical items organized by folders. SmartPlant Electrical defines standard folders in the database. You cannot move or delete these folders, nor can you add new folders. However, you can rename folders if required or create a custom folder inside a specific item type folder. Each folder contains a particular item type, and the options available on the menus depend on the currently selected item type.

Expand or collapse the hierarchy by clicking the + and **-** icons respectively or by double-clicking a folder.

List View Pane

The list view pane of the **Reference Data Explorer** displays items, other than folders, that are contained in the currently selected folder or item in the tree view pane.

Notes

- You can create several instances of the **Reference Data Explorer** by clicking **Window > New > Reference Data Explorer**.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics

Creating Reference Data Common Tasks

The following tasks are used frequently when you create reference electrical equipment such as motors, junction boxes, control stations, and cables in the **Reference Data Explorer**. Later on, you can use the reference equipment to generate specific items for your project in the **Electrical Index** with the minimum amount of manual data entry.

Shipped Reference Data

You create default data and values for each item type, for example motors, heaters, circuit breakers, or panels, in the Data Dictionary Manager. Default data affects all items at the plant level and specifies the values that appear when you create new items. Also, SmartPlant Electrical provides you with a number of predefined default items that appear in the **Reference Data Explorer** right after you install the application. For more information, see *Shipped Reference Data*, page 51.

Create a Plant Using Reference Data from an Existing Plant

In SmartPlant Engineering Manager, when you associate a new plant hierarchy with SmartPlant Electrical, if you already have existing plant data, you can specify an option to copy the reference data from an existing plant to the new plant. To do so, you first copy an existing plant structure, and then you load the reference data from the source plant. The copied data includes items from the **Reference Data Explorer** and some Options Manager settings. For more information, see *Create a Plant Using Reference Data from an Existing Plant*, page 52.

Create a Typical Electrical Motor

Most loads in your project are likely to be electrical motors, many with similar properties. For convenience of creating motors in the **Electrical Index**, you can create typical motors to use as templates for creating each group of project motors with common properties. Also, when you create an electrical motor, you can specify a space heater used to heat the motor windings during the non-operating state of the motor. For more information, see *Create a Typical Electrical Motor*, page 51.

Create a Typical Circuit

This feature allows you to create a typical circuit in the **Reference Data Explorer** to use it as a template for creating new project circuits that appear in the **Electrical Index**. While editing the common properties of a typical circuit, you can associate the circuit internal components with appropriate lookup tables. For more information, see *Create a Typical Circuit*, page 58.

Create a Reference Cable

You use reference cables as a basis for creating new cables in your project. Unlike other items, which you can create directly in the **Electrical Index** or copy from the **Reference Data Explorer**, you must create reference cables prior to creating new cables in the **Electrical Index**. For more information, see *Create a Reference Cable*, page 57.

Create Static Electrical Equipment

You can create the following types of static electrical equipment in your project:

- **Heaters** - Electrical process heaters are used for heating fluids in the plant.

- **Heat Traces** - An electrical heat trace is a kind of cable that is used to heat equipment and pipes, to ensure that the temperature for the process does not fall below the desired level.

- **Capacitors** - You can include power factor correction capacitors in your project.

- **Harmonic Filters** - A harmonic filter enables you to select and filter out a particular frequency in a power distribution network.

- **Resistors** - You can include power dissipation resistors in your project.

For more information, see *Create Static Electrical Equipment*, page 132.

Create Miscellaneous Electrical Equipment

Miscellaneous electrical equipment includes the following items that you can create in your project:

- **Lighting Fixtures** - You can add and define each lighting fixture in your plant, whether the fixture consists of single or multiple lighting units.

- **Socket Outlets** - You can add and define each socket outlet in your plant, whether the outlet consists of single or multiple sockets.

- **Welding Outlets** - You can add and define welding outlets in your plant.

- **Other Electrical Equipment** - You can add and define any other general type of electrical equipment in your plant.

For more information, see *Create Miscellaneous Electrical Equipment*, page 133.

Create a Generator

You can include generators to provide power supply for your utility. For more information, see *Create a Generator*, page 134.

Create a Battery Bank

You can include battery banks to provide a power supply backup for your utility. For more information, see *Create a Battery Bank*, page 135.

Create Converting Equipment

You can include the following types of converting electrical equipment in your project:

- **UPSs** - An Uninterrupted Power Supply (UPS) provides a means of power supply backup for your project.

- **Transformers** - You can include various transformers in your project wherever you need to step-down or step-up the voltage. You can create transformers with two or three sets of windings. For more information, see *Create a Transformer*, page 136.

- **Battery Chargers** - You can include battery chargers in your project to maintain your emergency batteries at optimum charge levels.

- **Variable Frequency Drives (VFDs)** - A variable frequency drive is a device that controls the speed of a motor by supplying power at different voltage / frequency ratios.

- **Other Converting Equipment** - Other converting equipment includes devices such as rectifiers.

For more information, see *Create Converting Equipment*, page 135.

Create a Custom Folder

This procedure allows you to create custom folders inside item type folders. Custom folders are used when you want to divide an item type into smaller specific groups according to parameters set by you. For example you could divide the generators folder into groups according to there output. For more information about creating a custom folder, see *Create a Custom Folder*, page 354.

Create Typical Schematic Blocks

Blocks are the basic graphical components of *schematics*. You create your library of blocks in the **Reference Data Explorer**. The definition of each block comprises the block name, the block type, and the drawing file associated with the block. You can create the drawing files for use with blocks in the Catalog Manager, or in an external CAD application such as SmartSketch, AutoCAD, or MicroStation. For more information, see:

- *Create Typical Schematic Blocks*, page 69.

- *Generate a Schematic*, page 282.

Create a Typical Schematic

Many electrical items have a common circuit layout, where the graphical components are identical, but the data is different. In order to generate a *schematic* for a particular load, you must create a typical schematic. A typical schematic includes drawing *blocks* and *macros* to represent the graphical components and data types respectively. Note that you can create schematic drawings that include several sheets. Multi-sheet schematics are mostly used when you need to display complex electrical equipment for which a single sheet is not enough to show all the relevant wiring details of the equipment. The software uses the typical schematic as a template for generating the schematic drawing that contains the actual load data. For more information, see:

- *Create a Typical Schematic*, page 71.
- *Create a Multi-Sheet Typical Schematic*, page 73.
- *Macro Definition Guidelines*, page 275.

Create a Profile

You can create a *profile* for an item in the **Reference Data Explorer** and use it to create associated items as well as to define default values for the main item and associated items in the **Electrical Index**. For more information, see *Create a Profile*, page 61.

Create Lookup Tables

Lookup tables enable you to define a set of default values for properties consisting of one or more key properties and their related common properties. For more information, see *Create Lookup Tables*, page 62.

Enable Additional Regional Standard Lookup Tables

SmartPlant Electrical allows you to enable regional standard lookup tables in addition to the NEC standards. You can define IEC and CEC standards in the Data Dictionary and then create lookup tables in the **Reference Data Explorer** based on these regional standards. You can then apply the new lookup tables to the appropriate equipment items that conform with these regional settings. For more information, see *Enable Additional Regional Standard Lookup Tables*, page 64.

Create Sets of Typical I/O Signals

You can create typical I/O signals grouped together in typical I/O sets. You can then include a typical I/O set in a profile that can be used with a project item or apply a typical I/O set to a project control station or a circuit. For more information about creating typical I/O signals, see *Create Typical I/O Signals*, page 74.

Associate and Apply a Lookup Table to a Circuit Internal Component

The software allows you to associate lookup tables with circuit internal components. You can make this association on the **Typical Circuit Common Properties** dialog box in the **Reference Data Explorer** and then apply that typical circuit to a number of project circuits. Alternatively, you can make this association on the **Circuit Common Properties** dialog box for a circuit that you select in the **Electrical Index** and then run the **Apply Lookup Tables** feature on the **Apply Options** dialog box to propagate the lookup table data to the circuit internal components. For more information, see *Associate and Apply a Lookup Table to a Circuit Internal Component*, page 68.

Automate the Sizing of Circuit Internal Components Based on Load Data

SmartPlant Electrical enables you to create lookup tables that include load properties as key fields.   You can associate these lookup tables with circuits or circuit internal components and apply these lookup tables to project circuits or circuit internals to populate the lookup table data. This feature is especially useful when creating protection equipment, fuses, circuit breakers, and so forth whose properties correlate with load lookup tables. For more information, see *Automate the Sizing of Circuit Internal Components Based on Load Data*, page 68.

Related Topics
- *Apply Lookup Table Data to Equipment*, page 143
- *Applying Options Common Tasks*, page 141
- *Populating Items with Data: An Overview*, page 140
- *Shipped Reference Data*, page 51

Shipped Reference Data

SmartPlant Electrical provides with a number of predefined default items that appear in the **Reference Data Explorer** right after you install the application. These items are provided to you as samples that you can use to create other default or project items.

The following reference items are available:

- Typical motors

- Profiles for motors

- Look-up tables for motors

- Control stations with associated control cables

- Typical power, control, and ground cables. (Note that if you want to use the shipped reference data provided with the application, please validate that these values are appropriate for cable sizing. Note that shipped reference data is provided to you as uncertified samples only.)

- Typical circuits, including converting equipment of all types other than 3-winding transformers

- Typical schematic blocks

- Typical schematic drawings

Notes

- The shipped reference data uses American regional settings; in other countries which use different regional settings, you have to replace the supplied reference data with data that was created using the appropriate regional settings, to ensure that the software uses the correct standard values for performing calculations.

- In addition to using the shipped standard reference data, you can populate the plant with reference data that you can download from a network location using the Options Manager. The download contains the following reference items:

- Lookup tables containing typical IEC motor data

- Lookup tables containing typical ANSI motor data

- Lookup tables containing NEC motor data with appropriate FLA and LRC values (NEC tables 430-7b, 430-147 through 151)

- IEC reference cables

- ANSI reference cables

Create a Plant Using Reference Data from an Existing Plant

🔷 **Important**

- At least one plant that is already associated with SmartPlant Electrical for use as your source must exist in the **Plant hierarchy** tree.

1. In SmartPlant Electrical, open the source plant and create all the reference data that you want to copy (profiles, lookup tables, reference items, and so forth).

2. In the Options Manager, make the following definitions for the source plant: Formats, Item Type Definitions, Naming Conventions, General Settings, Plant Default Preferences, and Project-Wide Parameters.

   💡 **Tip**

   - The software sets the defaults for the locations in the target plant according to the reference data path that you specify when copy the data (see step 12).

3. In the Options Manager, click **Options** > **Naming Conventions** > **Copy to Reference Data**.

   🔷 **Important**

   - You must perform this step if you want to the naming conventions to be available for copying when you associate the target plant with SmartPlant Electrical.

4. Open SmartPlant Engineering Manager.

5. In the left pane (tree view), select the plant that you want to use as the source plant.

6. In the right pane, select the application, right-click, and click **New Data Dictionary Template**.

7. Navigate to the path where you want to create a template (.ddt) file containing the source reference data and type a name for the file.

8. In the left pane, create a new plant or select an existing plant that is not associated with an application. This will be your target plant.

9. Right-click the **Applications** sub-folder for the target plant and click **Associate Applications**.

   💡 **Tips**

   - If necessary, expand the plant folder to display the **Applications** sub-folder.

   - For details of the **Associate Applications** wizard parameters, refer to SmartPlant Engineering Manager Online Help.

10. Under **Data dictionary source**, click **Use custom template**.

11. Beside **Template path**, click the ellipsis button ⬚ to navigate to the .ddt file that you created.

12. To copy values to Options Manager Locations, on the wizard page with the title **Enter the database server and user connection information for SmartPlant Electrical schema**, enter a path in the **SmartPlant Electrical reference data path** box.

13. On the wizard page with the title **Enter the database server and user connection information for SmartPlant Electrical reference schema**, select the check box **Load reference data from <plant name>**.

14. Complete the remaining pages of the wizard with the information that you require.

15. In the Options Manager, copy the naming conventions from the seed plant. For full details of how to copy naming conventions, see the Options Manager User's Guide: *Copy Naming Conventions for Plant Group*.

Related Topics
- *Creating Reference Data Common Tasks*, page 46.
- *Working with the Reference Electrical Engineer: An Overview*, page 92.

Create a Reference Item

1. In the **Reference Data Explorer**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Right-click the folder or parent item where you are going to add the new item, and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the item.

4. Edit the item properties in the **Properties** window according to the catalog information.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create the item.

- Some items have a **Common Properties** dialog box from which you can edit the most common properties for the item. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the item. For full details of the properties, see the appropriate dialog box topic.

- *Reference Cable Common Properties Dialog Box*, page 646

Create a Typical Electrical Motor

1. In the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Motors**.

2. Right-click the **Motors** folder, and then click **New Motor** or **New Motor with Heater**.

3. Accept the default item tag or type a name of up to 80 characters for the motor.

4. Right-click the motor and click **Common Properties** to open the **Electrical Motor Common Properties** dialog box.

5. On the **Electrical Motor Common Properties** dialog box, edit the motor properties as required. For more details, see *Edit Electrical Motor Common Properties*, page 129.

> ✎ **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new electrical motor.

- The **Electrical Motor Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new electrical motor.

- You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

- When you add a motor with a space heater, the heater appears as a sub-item of the motor. If you subsequently delete the motor, the software also deletes the heater.

Related Topics
- *Creating Reference Data Common Tasks*, page 46.
- *Delete an Item*, page 40
- *Duplicate an Electrical Motor*, page 130
- *Edit Electrical Motor Common Properties*, page 129
- *Rename an Item*, page 39

Creating Typical Panels

You can create the following types of typical panels in the **Reference Data Explorer**:

- **Control Stations** - Small cabinets containing low voltage start-stop circuitry used to control a load at a remote location. For more information, see *Create a Typical Control Station*, page 55.

- **Local Panels** - Cabinets containing circuit, equipment, and wiring items, used for example, to provide switching or isolation close to a motor. For more information, see *Create a Typical Local Panel*, page 56.

- **Junction Boxes** - Cabinets used to connect wires that run between various items of electrical equipment. For more information, see *Create a Typical Junction Box*, page 57.

Note

- In contrast to *power distribution boards (PDBs)*, panels are much smaller in size and do not contain *busbars*.

Related Topics
- *Creating Project Panels*, page 175
- *Creating Reference Data Common Tasks*, page 46
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38

Create a Typical Control Station

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Control Stations** folder, and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the control station.

   ### Tip

   - If you are creating a number of typical control stations at one time, you may want to stop at this point and edit the typical control station properties at a later stage.

4. On the **Reference Control Station Common Properties** dialog box, edit the typical control station properties as required. For more details, see *Edit Typical Control Station Common Properties*, page 55.

Notes

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new typical control station.

- You access the **Reference Control Station Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new typical control station.

Related Topics
- *Create a Control Station*, page 176
- *Creating Typical Panels*, page 52
- *Edit Typical Control Station Common Properties*, page 55

Edit Typical Control Station Common Properties

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Panels** > **Control Stations**.

2. Right-click the desired control station and on the shortcut menu, click **Common Properties**.

3. On the **Reference Control Station Common Properties** dialog box, add a cable to the control station by clicking **Add** and performing the following steps.

4. On the **Select Reference Cable** dialog box, select the desired cable specification.

5. Click **Find** to open the **Reference Cable Library** dialog box, and select the cable that you require.

6. Repeat the preceding steps if you require additional cables.

7. To remove a cable from the control station, select the desired cable under **Associated cables** and click **Remove**.

Note

- You can define as many reference control stations as required for assigning to loads by applying equipment profiles.

Related Topics
- *Create a Control Station*, page 176
- *Creating Reference Data Common Tasks*, page 46
- *Creating Typical Panels*, page 52

Create a Typical Local Panel

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Local Panels** folder and then click **New Local Panel**.

3. Accept the default item tag or type a name of up to 80 characters for the local panel.

Related Topics
- *Creating Reference Data Common Tasks*, page 46
- *Creating Typical Panels*, page 52

Create a Typical Junction Box

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Junction Boxes** folder and then click **New Junction Box**.

3. Accept the default item tag or type a name of up to 80 characters for the junction box.

Related Topics
- *Creating Typical Panels*, page 52

Create a Reference Cable

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Right-click the **Power**, **Control**, **Grounding**, or **Instrumentation** folder according to the type of cable required, and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the cable.

4. Enter values for the cable properties on the **Reference Cable Common Properties** dialog box according to the available catalog information. You must enter values for those properties marked with an asterisk.

   💡 **Tip**

   - To enable sizing of a power cable in the project, you should enter values for all the properties on the **General** tab.

📝 Notes

- You access the **Reference Cable Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the cable.

- Short circuit sizing calculations for project power cables use data in the **Conductors** lookup tables, and this data is then used to select the appropriate reference cable. For this reason, when you create a new reference cable, it is recommended that you open the **Data** tab of each lookup table to ensure that appropriate data related to the reference cable is available, and if it is not, to add the data on the lookup table.

Related Topics
- *Creating Reference Data Common Tasks*, page 46
- *Edit Reference Cable Common Properties*, page 57
- *Reference Cable Common Properties Dialog Box*, page 646

Edit Reference Cable Common Properties

1. In the **Reference Data Explorer**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand the desired cable folder: **Power Cables**, **Control Cables**, or **Grounding Cables**.

3. Right-click the desired cable and on the shortcut menu, click **Common Properties**.

4. Enter values for the cable properties on the **Reference Cable Common Properties** dialog box according to the available catalog information. You must enter values for those properties marked with an asterisk.

   ### 💡 Tip

   - To enable sizing of a power cable in the project, you should enter values for all the properties on the **General** tab.

📝 **Notes**

- You can edit the properties of a reference cable as long as it has not been used in the **Electrical Index** with any project item or if the reference cable is not part of any existing profile or control station in the **Reference Data Explorer**.

- Short circuit sizing calculations for project power cables use data in the **Conductors** lookup tables, and this data is then used to select the appropriate reference cable. For this reason, if you change the values of reference cable properties, it is recommended that you open the **Data** tab of each lookup table to ensure that appropriate data related to the reference cable is available, and if it is not, to add the data on the lookup table.

Related Topics
- *Create a Reference Cable*, page 57
- *Creating Reference Data Common Tasks*, page 46

Create a Typical Circuit

1. In the **Reference Data Explorer**, right-click the **Typical Circuits** folder and then click an option to create a new typical circuit of one of the following standard types:

   - **New Feeder Circuit** 🔲
   - **New Incomer Circuit** 🔲
   - **New** *Coupler Circuit* 🔲
   - **New** *Bus Riser Circuit* 🔲
   - **New** *Panel Circuit* 🔲

2. Accept the default item tag or type a name of up to 80 characters for the circuit.

3. Right-click the circuit and on the shortcut menu, click an option to create a new internal component.

   💡 **Tip**

   - You can also create a circuit component by dragging an item from a folder under **Disconnect Electrical Equipment**. For more information, see *Copy Components to a Typical Circuit*, page 59.

4. Right-click the new typical circuit and open its common properties.

5. In the **Details of circuit internals** data window, click under **Lookup Table** and select an appropriate lookup table that you want to associate with the corresponding circuit internal component.

💡 **Tip**

- You must first create an appropriate lookup table that is associated with the current circuit internal component. For details, see *Associate and Apply a Lookup Table to a Circuit Internal Component*, page 68.

6. Repeat the previous steps as often as needed to create additional items for the circuit.

Related Topics
- *Create a Typical Circuit*, page 58
- *Create Circuit Components*, page 173
- *Creating Reference Data Common Tasks*, page 46

Copy Components to a Typical Circuit

1. In the **Reference Data Explorer**, expand the **Typical Circuits** folder and locate the desired typical circuit.

   💡 **Tip**

   - Before dragging items between folders in the **Reference Data Explorer**, for convenience, you can create another **Reference Data Explorer** window. To do this, click **Window** > **New** > **Reference Data Explorer**.

2. Expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Disconnect Electrical Equipment**.

3. In one of the following folders, expand the folder and select an existing item, or create a new item in the folder:

   - **Contactors**

   - **Overload Relays**

   - **Disconnect Switches**

   - **Circuit Breakers**

   - **Fuses**

   - **Starters**

   - **Other Disconnect Equipment**

   - **Metering Equipment** (select a sub-folder)

4. Drag the desired item to the typical circuit.

5. Repeat the above steps as often as needed to create additional components for the circuit.

Related Topics
- *Create a Typical Circuit*, page 58
- *Create Circuit Components*, page 173

Copy from Typical Circuit Components

1. In the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Disconnect Electrical Equipment**.

   💡 **Tip**

   - Before dragging items between folders in the **Reference Data Explorer**, for convenience, you can create another **Reference Data Explorer** window. To do this, click **Window** > **New** > **Reference Data Explorer**.

2. In the **Typical Circuits** folder, expand a typical circuit.

3. Drag the desired circuit component in the typical circuit to the **Disconnect Electrical Equipment** sub-folder appropriate for the item type from among the following:

   - **Contactors**
   - **Overload Relays**
   - **Disconnect Switches**
   - **Circuit Breakers**
   - **Fuses**
   - **Starters**
   - **Other Disconnect Equipment**
   - **Metering Equipment** (select a sub-folder)

   📝 **Note**

   - After copying an item by dragging it from a typical circuit to a **Disconnect Electrical Equipment** sub-folder, you can drag the item to copy it to another typical circuit.

Related Topics
   - *Copy Components to a Typical Circuit*, page 59
   - *Create Circuit Components*, page 173

Create a Profile

1. In the **Reference Data Explorer**, expand the **Profiles** folder hierarchy by clicking the + icon and select the item type folder for the profile that you want to create.

2. Right-click the folder and then click **New Profile**.

3. Accept the default item tag or type a name of up to 80 characters for the profile.

   💡 **Tip**

   • If you are creating a number of profiles at one time, you may want to stop at this point and edit the profile properties at a later stage.

4. On the **Equipment Profile Properties** dialog box, edit the profile properties as required. For more details, see *Edit Profile Common Properties*, page 62.

   📝 **Notes**

   • On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new profile.

   • You access the **Equipment Profile Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new profile.

   Related Topics
   • *Apply a Profile to an Item*, page 147
   • *Creating Reference Data Common Tasks*, page 46
   • *Edit Profile Common Properties*, page 62

Edit Profile Common Properties

1. In the **Reference Data Explorer**, expand the **Profiles** folder hierarchy by clicking the + icon and select the item type folder for the profile that you want to edit.

2. Right-click the desired profile and on the shortcut menu, click **Common Properties**.

3. On the **Equipment Profile Properties** dialog box, click the **General** tab.

4. To change the profile name, click in the **Profile tag** box and type the profile name or click the ellipsis button ⬚ beside the box.

   💡 **Tip**

   • If you want to use this profile as the default whenever you apply a profile to an item, select **Set as default**.

5. Under **Default reference item**, select a reference item for copying the property values when you apply the profile.

   💡 **Tip**

   - If you choose a motor that includes an internal heater, the **Space Heater** tab becomes enabled.

6. To associate cables with the item referenced in the profile, under **Associated cables**, click **Add** to open the **Select Reference Cable** dialog box, where you can select the desired cable.

   💡 **Tip**

   - You can add more than one cable to a profile.

7. Under **Typical control station**, click **Add**.

8. On the **Find** dialog box, click **Find Now** to display the available control stations.

9. In the list of control stations, highlight each control station that you want to associate with the profile.

10. Click **OK** to add the selected control stations to the profile and return to the **Equipment Profile Properties** dialog box.

11. Click the **Protection Devices** tab.

12. Under Typical circuit, click **Find** to find and select a typical circuit.

    💡 **Tip**

    - After selecting a typical circuit, the software displays the circuit function and the existing internal components.

13. For a motor that includes an internal heater, click the **Space Heater** tab and define cables and a typical circuit as required for the heater.

14. Click the **Miscellaneous** tab to specify a typical schematic and lookup table for the profile.

15. Under **Typical schematic**, click **Find** to find and select a typical schematic.

16. Under **Lookup table**, click **Find** to find and select a lookup table.

📝 **Note**

- For more details of how to find items, see the *Find dialog box*, page 580.

Create Lookup Tables

1. In the **Reference Data Explorer**, expand the **Lookup Tables** folder by clicking the + icon beside the folder.

2. Right-click the folder for the item that you require and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the item lookup table.

   💡 **Tip**

   - If you are creating a number of lookup tables at one time, you may want to stop at this point and edit the lookup table properties at a later stage.

4. On the **Lookup Table Properties** dialog box, edit the lookup table properties as required. For more details, see *Edit Lookup Tables*, page 65.

📝 **Notes**

   - On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new lookup table.

   - You access the **Lookup Table Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new lookup table.

Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Apply Lookup Table Data to Equipment*, page 143
   - *Creating Reference Data Common Tasks*, page 46
   - *Edit Lookup Tables*, page 65
   - *Lookup Table Properties Dialog Box*, page 644

Enable Additional Regional Standard Lookup Tables

1. In the Data Dictionary, add a new regional standard entry in the **Conformity to standard** select list.

2. Switch to SmartPlant Electrical and in the **Reference Data Explorer**, create a new lookup table for the regional standard entry that you just created in the Data Dictionary.

3. Apply the new lookup table to all the equipment items in your project that conform with the new regional standard.

   Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Apply Lookup Table Data to Equipment*, page 143
   - *Creating Reference Data Common Tasks*, page 46
   - *Edit Lookup Tables*, page 65
   - *Lookup Table Properties Dialog Box*, page 644

Edit Lookup Tables

1. In the **Reference Data Explorer**, expand the **Lookup Tables** folder by clicking the + icon beside the folder.

2. Expand the lookup table folder for the desired item type.

3. Right-click the desired lookup table and on the shortcut menu, click **Common Properties**.

4. On the **Lookup Table Properties** dialog box, click the **Structure** tab to determine the item properties that you want to use for applying data.

   💡 **Tip**

   - To define the lookup table as the default for the current item type, select **Set as default**.

5. Click **Add** to add a new property line to the lookup table definition.

6. Under **Edit**, select in the **Property** list the desired property to use for the lookup table.

   🔻 **Caution**

   - It is good practice not to include calculated properties in lookup tables. For example, if the **Supply AC/DC Flag** property for a motor is set to **DC**, and you try to select a value for the **Frequency** property, you cannot apply this value and the software rejects the operation. You must first set **Supply AC/DC Flag** to **AC**, and only then you can apply the lookup table value for the **Frequency** property to the motor.

7. If required, modify the caption text for the header that appears on the **Data** tab. The default is the property name.

8. From **Input-output value**, select **Input** if you want to define the property as a key field - otherwise, select **Output**.

   💡 **Tips**

   - The software uses key field values to identify the data to update using the appropriate values for the output fields.

   - The software treats read-only data as input only, and the **Input-output value** property is disabled for such data.

9. From **Overwrite option**, select **True** if you want to overwrite existing data when you apply the lookup table values, or select **False** to insert the values only where no data exists previously.

10. Repeat the preceding steps for as many other properties as you require.

11. If required, change the order of the properties under **Definition** by selecting an individual property and clicking **Up** or **Down**.

   💡 **Tip**

   - The order of the properties affects how they are arranged on the **Data** tab. It is usually convenient to arrange the input properties at the beginning, followed by the output properties. For more details, see *Lookup Table Structure Example*, page 65.

12. Click the **Data** tab and enter data values for the desired properties.

13. Enter data values for the properties that you require on the blank line.

   💡 **Tips**
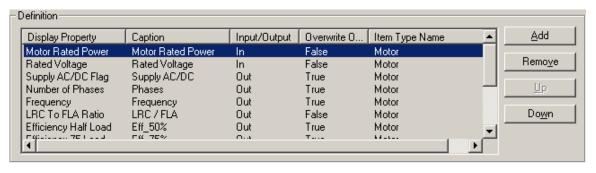
   - You can right-click individual cells to copy and paste data. To copy and paste entire rows of data, select the source row and click **Copy Row**. Next, select the target row and click **Paste Row**.

   - Ensure that each combination of data in the key fields is unique. If you enter duplicate key fields in two rows, the software displays a warning message and highlights the duplicate row so that you can modify he input data. For more details, see *Lookup Table Data Example*, page 67.

   - You can sort the rows of data according to a particular column by clicking the column header. Repeatedly clicking a column header displays the data in alternating ascending and descending order.

14. Repeat the previous step for each set of data required.

✎ Notes

- You can define as many lookup tables as you require for a particular item type.

- There are three specific lookup tables for conductors that cannot be deleted: **Conductor Material Coefficients**, **Conductor Insulation Temperatures**, and **Conductor Area cmils**. These tables contain typical values for some of the most common conductor materials and conductor insulation materials. You should check these typical values and make sure they comply with and match the values used or specified by local codes or cable manufacturers. The software uses the data in these lookup tables for short circuit cable sizing and you should add appropriate data for any additional conductor material or insulation if needed.
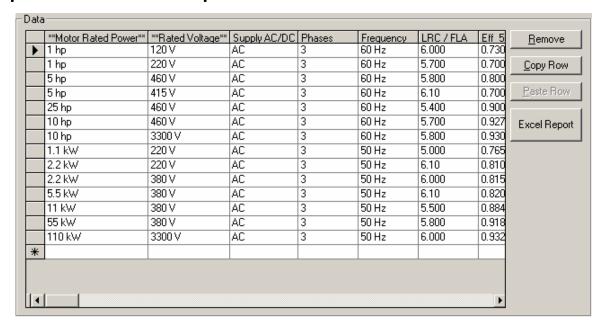
Related Topics
- *Apply Lookup Table Data to Equipment*, page 143

Lookup Table Structure Example

Lookup Table Data Example

	Motor Rated Power	**Rated Voltage**	Supply AC/DC	Phases	Frequency	LRC / FLA	Eff_5	
▶	1 hp	120 V	AC	3	60 Hz	6.000	0.730	Remove
	1 hp	220 V	AC	3	60 Hz	5.700	0.700	Copy Row
	5 hp	460 V	AC	3	60 Hz	5.800	0.800	
	5 hp	415 V	AC	3	60 Hz	6.10	0.700	Paste Row
	25 hp	460 V	AC	3	60 Hz	5.400	0.900	
	10 hp	460 V	AC	3	60 Hz	5.700	0.927	Excel Report
	10 hp	3300 V	AC	3	60 Hz	5.800	0.930	
	1.1 kW	220 V	AC	3	50 Hz	5.000	0.765	
	2.2 kW	220 V	AC	3	50 Hz	6.10	0.810	
	2.2 kW	380 V	AC	3	50 Hz	6.000	0.815	
	5.5 kW	380 V	AC	3	50 Hz	6.10	0.820	
	11 kW	380 V	AC	3	50 Hz	5.500	0.884	
	55 kW	380 V	AC	3	50 Hz	5.800	0.918	
	110 kW	3300 V	AC	3	50 Hz	6.000	0.932	
*								

Associate and Apply a Lookup Table to a Circuit Internal Component

1. In the **Reference Data Explorer**, expand the **Lookup Tables** folder by clicking the + icon beside the folder.

2. Right-click the folder for a circuit internal component (for example, **Circuit Breakers**, **Fuses**, and so on) and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the item lookup table.

   💡 **Tip**

   - If you are creating a number of lookup tables at one time, you may want to stop at this point and edit the lookup table properties at a later stage.

4. On the **Lookup Table Properties** dialog box, edit the lookup table properties as required. For more details, see *Edit Lookup Tables*, page 65.

5. Create or edit the properties of a typical circuit in the **Reference Data Explorer**. For details, see *Create a Typical Circuit*, page 58.

6. On the **Typical Circuit Common Properties** dialog box, under **Details of circuit internals**, select an appropriate lookup table from **Lookup Table** list.

7. In the **Electrical Index**, select a project circuit to which you want to apply a lookup table.

8. Click **Tools** > **Apply Options**.

9. On the **Apply Options** dialog box, click the **Lookup Tables** tab.

10. Select the **Enable associated lookup tables of circuit internals** check box.

11. Click **Apply**.

☑ Notes

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new lookup table.

- You access the **Lookup Table Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new lookup table.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply Lookup Table Data to Equipment*, page 143
- *Creating Reference Data Common Tasks*, page 46
- *Edit Lookup Tables*, page 65
- *Lookup Table Properties Dialog Box*, page 644

Automate the Sizing of Circuit Internal Components Based on Load Data

1. In the **Reference Data Explorer**, expand the **Lookup Tables** folder by clicking the + icon beside the folder.

2. Right-click the folder for a circuit internal component (for example, **Circuit Breakers**, **Fuses**, and so on) and then click **New**.

3. Accept the default item tag or type a name of up to 80 characters for the item lookup table.

   ### 💡 Tip

   - If you are creating a number of lookup tables at one time, you may want to stop at this point and edit the lookup table properties at a later stage.

4. On the **Lookup Table Properties** dialog box, edit the lookup table properties as required. For more details, see *Edit Lookup Tables*, page 65.

5. Create or edit the properties of a typical circuit in the **Reference Data Explorer**. For details, see *Create a Typical Circuit*, page 58.

6. On the **Typical Circuit Common Properties** dialog box, under **Details of circuit internals**, select an appropriate lookup table from **Lookup Table** list.

7. In the **Electrical Index**, select a project circuit to which you want to apply a lookup table.

8. Click **Tools** > **Apply Options**.

9. On the **Apply Options** dialog box, click the **Lookup Tables** tab.

10. Select the **Enable associated lookup tables of circuit internals** check box.

11. Click **Apply**.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new lookup table.

- You access the **Lookup Table Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new lookup table.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply Lookup Table Data to Equipment*, page 143
- *Creating Reference Data Common Tasks*, page 46
- *Edit Lookup Tables*, page 65
- *Lookup Table Properties Dialog Box*, page 644

Create Typical Schematic Blocks

1. In the **Reference Data Explorer**, right-click the **Typical Blocks** folder, and then click **New**.

2. Accept the default item tag or type a name of up to 50 characters for the typical block.

   💡 **Tip**

   - If you are creating a number of schematic blocks at one time, you may want to stop at this point and edit the schematic block properties at a later stage.

3. On the **Typical Schematic Block Common Properties** dialog box, edit the schematic block properties as required. For more details, see *Edit Typical Schematic Block Common Properties*, page 70.

✎ **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new typical schematic block.

- You access the **Typical Schematic Block Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new typical schematic block.

Related Topics
- *Creating Reference Data Common Tasks*, page 46
- *Edit Typical Schematic Block Common Properties*, page 70
- *Working with the Reference Data Explorer: An Overview*, page 43

Edit Typical Schematic Block Common Properties

1. In the **Reference Data Explorer**, expand the **Typical Blocks** folder by clicking the + icon beside the folder.

2. Right-click the desired block and on the shortcut menu, click **Common Properties**.

3. On the **Typical Schematic Block Common Properties** dialog box, under **Block details**, select the desired block type in the **Block type** list.

4. Beside **File name**, click the ellipsis button ⸬ to navigate to the drawing file that contains the graphical representation of the block.

   💡 **Tip**

   - You can only select drawing files from the default folder specified in the Options Manager; you can navigate to other folders, but you will not be able to select any files in those folders.

5. Click **Add** to add the block to the library without having to close the dialog box.

6. To delete from the database a block that is not used in any typical schematic, select the desired block and click **Delete**.

7. When done, click **Apply**.

✎ **Note**

- Each block can have one or more associated *macros*. You can define and edit macros using the Catalog Manager or an external CAD application such as SmartSketch, AutoCAD, or MicroStation.

Create a Typical Schematic

1. In the **Reference Data Explorer**, right-click the **Typical Schematics** folder, and then click **New**.

2. Accept the default item tag or type a name of up to 80 characters for the typical schematic.

   💡 **Tip**

   - If you are creating a number of typical schematics at one time, you may want to stop at this point and edit the typical schematic properties at a later stage.

3. On the **Typical Schematic Common Properties** dialog box, edit the typical schematic properties as required. For more details, see *Edit Typical Schematic Common Properties*, page 72.

   📝 **Notes**

   - On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new typical schematic.

   - You access the **Typical Schematic Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new typical schematic.

   - You can add various macros to the title block that will appear in the schematic drawing. For more details, see *Title Block Macros*, page 267.

Related Topics
- *Create a Multi-Sheet Typical Schematic*, page 73
- *Creating Reference Data Common Tasks*, page 46
- *Edit Typical Schematic Common Properties*, page 72
- *Title Block Macros*, page 267

Edit Typical Schematic Common Properties

1. In the **Reference Data Explorer**, expand the **Typical Schematics** folder by clicking the + icon beside the folder.

2. Right-click a schematic and on the shortcut menu, click **Common Properties**.

3. On the **Typical Schematic Common Properties** dialog box, add a block to the schematic by clicking **Add** to add a new line to the list under **Schematic blocks**.

   ◆ **Important**

   • When adding blocks that comprise the typical schematic, make sure that all the blocks belong to the same CAD format. Generating a schematic drawing based on a typical schematic that uses different CAD formats (for example, .sym and .dgn files) may result in generation errors.

4. Under **Block details**, select the desired block in the **Block name** list.

   💡 **Tip**

   • You can click **View** to preview the currently selected block before adding it to the typical schematic.

5. Select the starting point for the bottom left corner of the block relative to the bottom left corner of the schematic by specifying the values in the **X** and **Y** boxes. These are usually the same as the coordinates that were originally defined for the block while it was a part of the complete schematic. The default values for the coordinates are 0,0.

6. For the **Sequence** property, if needed, type a positive integer to represent the sequence in which the software associates the blocks with matching project items.

   💡 **Tip**

   • The software uses these sequence numbers if there are multiple project items belonging to the same item type, for example, a schematic that contains two control station blocks that have to be associated with project items. The software associates the block that has lowest sequence number with the plant item that has the lowest **Sequence in group** attribute among the project items belonging to same item type. Example: A typical schematic contains two blocks A and B whose sequence numbers are 5 and 8 respectively. If used with a motor that has two control stations CS-1 and CS-2 for which the **Sequence in group** properties are 3 and 8 respectively, the software matches Block A with Control Station CS-1 and Block B with Control Station CS-2.

7. Set the number of sheets in the typical schematic and define the sheet properties. For details, see *Create a Multi-Sheet Typical Schematic*, page 73.

8. Under **Sheet number**, select a sheet that you want to assign to the selected block.

💡 **Tips**

- You can assign the same block to several different sheets if you want the block to appear more than once on different sheets of the same schematic.

- Note that the software does not retain in the database the sheets that have not been assigned to blocks.

9. To remove a block from the schematic, select the desired block under **Schematic blocks** and click **Remove**.

10. When done, click **Apply**.

Related Topics
- *Create a Multi-Sheet Typical Schematic*, page 73
- *Creating Reference Data Common Tasks*, page 46
- *Edit Item Properties*, page 38

Create a Multi-Sheet Typical Schematic

1. On the **Typical Schematic Common Properties** dialog box, click **Sheets**. For more details on how to open the **Typical Schematic Common Properties** dialog box, see *Edit Typical Schematic Common Properties*, page 72.

2. On the **Schematic Sheet Properties** dialog box, click **Add**.

3. Under **Sheet number** in the **Sheet details** group box, type the sheet sequence number.

4. Under **Description** in the **Sheet details** group box, type the sheet description.

5. Click **Add** again to add another sheet to the same typical schematic and repeat steps 3 and 4.

6. Click **OK** to return to the **Typical Schematic Common Properties** dialog box and assign a sheet number to each block in the typical schematic.

📝 **Notes**

- You can assign the same block to several different sheets if you want the block to appear more than once on different sheets of the same schematic.

- You should assign a sheet to each block in the typical schematic. Unassigned sheets are not kept in the database by the software.

Related Topics
- *Create a Typical Schematic*, page 71
- *Edit Typical Schematic Common Properties*, page 72

Create Typical I/O Signals

1. In the **Reference Data Explorer**, right-click the **Typical I/O sets** folder, and then click **New Typical I/O Set**.

2. Right-click the new typical I/O set and select **New Signal** on the short-cut menu.

3. Accept the default item tag or type a name of up to 80 characters for the I/O signal.

4. Enter values for the I/O signal in the **Properties** window as you require.

Related Topics
- *Creating Reference Data Common Tasks*, page 46
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38
- *Edit Typical Schematic Block Common Properties*, page 70
- *Rename an Item*, page 39
- *Working with the Reference Data Explorer: An Overview*, page 43

Working with the Electrical Index: An Overview

You use the **Electrical Index** for creating items in your SmartPlant Electrical inventory, selecting them for use in your electrical configuration, and managing assets with edit, duplicate, and delete operations.

The main features of the **Electrical Index** are:

Toolbar

The toolbar contains the following options:

Find - Click **Find** to open the **Find** dialog box where you can look for a specific item in the **Electrical Index**. Click the arrow to display a menu with the following options:

- **Find** - Opens the **Find** dialog box where you can look for a specific item in the **Electrical Index**.

- **Find in Electrical Engineer** - Searches the **Electrical Engineer** for the item that is currently selected in the **Electrical Index**. Note that both the **Electrical Engineer** and the **Electrical Index** have to be open if you want to use this feature.

- **Close** - Closes the active **Electrical Index** window.

Refresh - Click **Refresh** to update the **Electrical Index** display. This feature is useful where multiple users are working on the same set of data.

Views - The **Views** icon allows you to toggle through different views of the items in the list view pane. The arrow beside the icon allows you to select a specific view.

Buttons - Buttons provide a useful shortcut to an item or a group of items. The **Buttons** icon allows you to toggle between showing and hiding existing buttons. The arrow beside the icon includes options that allow you to add or remove buttons for the currently selected item or folder. Note that when removing a button, if you do not select a specific button, the software removes the uppermost button in the list.

Tree View Pane

The tree view pane of the **Electrical Index** shows the equipment in your inventory organized by folders. SmartPlant Electrical defines standard folders in the database. You cannot move or delete these folders, nor can you add new folders. However, you can rename folders if required or create a custom folder inside a specific item type folder. Each folder contains a particular item type, and the options available on the menus depend on the currently selected item type.

Expand or collapse the hierarchy by clicking the **+** and **-** icons respectively or by double-clicking a folder.

When you select a folder, the number of items in the folder appears in the status bar.

List View Pane

The list view pane of the **Electrical Index** displays items, other than folders, that are contained in the currently selected folder or item in the tree view pane.

Notes

- You can create several instances of the **Electrical Index** by clicking **Window > New > Electrical Index**.

- You can right-click any item in the **Electrical Index** to find it in the **Electrical Engineer** or click an item in the **Electrical Engineer** to find it in the **Electrical Index**. Both the **Electrical Engineer** and the **Electrical Index** have to be open if you want to use this feature.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics

Working with the Electrical Engineer: An Overview

You use the **Electrical Engineer** to manage electrical associations among equipment in your plant. You can make new electrical associations or change the existing ones. The **Electrical Engineer** displays electrical items based on the electrical relationships among them, for example source to load. The **Electrical Engineer** also shows every item that has an electrical connection with a selected item. The **Electrical Engineer** allows you to generate and display single line diagrams (SLDs) that show the electrical network distribution powered from an item selected in the **Electrical Engineer**.

The software synchronizes any new electrical relationships that you create in the **Electrical Engineer** with the **Electrical Index**. When creating a new item in the **Electrical Index**, the software also adds this item in the **Electrical Engineer** if appropriate (some items do not appear the **Electrical Engineer**).

Display Rules

The **Electrical Engineer** displays electrical items based on their electrical relationship. Therefore, a power source, for example, is always on a higher hierarchical level than a load. The following rules define which electrical items appear in the **Electrical Engineer**:

- The **Electrical Engineer** displays all electrical items that have at least one electrical association.

- Certain items appear automatically in the **Electrical Engineer** after you create them in the **Electrical Index**. These items appear in the **Electrical Engineer** even if they do not have any electrical association. These items are generators, buses, circuits, battery banks, battery chargers, UPSs, and transformers.

- The **Electrical Engineer** automatically displays power cables that the software creates when you create a load or when you activate the **Apply Option** command to automatically associate that load with a feeder.

- For a multiple cable assembly, you can hide or display the individual cables by right-clicking the icon , and on the shortcut menu, clicking **Show Multiple Cables**. The software indicates the display toggle state by a check mark.

- Items that have no hierarchical parents appear at the root of the **Electrical Engineer** provided that they feed other items.

- Certain items do not appear in the **Electrical Engineer**. These items are power distribution boards, cells, control stations, and their cables. Only the items that are included in single line diagrams or power distribution networks appear in the **Electrical Engineer** except for circuit components that do not appear in the **Electrical Engineer** at all.

- You can display item tags of buses, bus risers, and incomers together with the item tags of the associated power distribution boards if you set your general preferences accordingly.

The main features of the **Electrical Engineer** are:

Toolbar

The toolbar contains the following options:

Find - Click **Find** to open the **Find** dialog box where you can look for a specific item in the **Electrical Engineer**. Click the arrow to display a menu with the following options:

- **Find** - Opens the **Find** dialog box where you can look for a specific item in the **Electrical Engineer**.

- **Find in Electrical Index** - Searches the **Electrical Index** for the item that is currently selected in the **Electrical Engineer**. Note that both the **Electrical Engineer** and the **Electrical Index** have to be open if you want to use this feature.

- **Close** - Closes the active **Electrical Engineer** window.

Refresh - Click **Refresh** to update the **Electrical Engineer** display. This feature is useful where multiple users are working on the same set of data.

Buttons - Buttons provide a useful shortcut to an item or a group of items. Click **Buttons** to toggle show / hide for existing buttons. To create a button for the currently selected item or folder, click the arrow and click **Add Button**. To remove a button, select the button to remove, then click the arrow, and click **Remove Button**. Note that if you do not select any button to remove, the software removes the uppermost button in the list.

Tree View Pane

The tree view pane of the **Electrical Engineer** shows electrical items organized in the **Plant** folder. You cannot move, delete or rename this folder, nor can you add new folders. Each item contained in the **Plant** folder contains other items that are associated with it electrically. The options available on the menus depend on the currently selected item type.

Expand or collapse the hierarchy by clicking the + and - icons respectively or by clicking an item in the **Plant** folder hierarchy.

 Notes

- You can open several instances of the **Electrical Engineer** by clicking **Window > New > Electrical Engineer**.

- You can right-click any item in the **Electrical Index** to find it in the **Electrical Engineer** or click an item in the **Electrical Engineer** to find it in the **Electrical Index**. Both the **Electrical Engineer** and the **Electrical Index** have to be open if you want to use this feature.

- You can right-click any item in the **Electrical Engineer** and then click **Expand All Sub-Items** on the shortcut menu to display all the existing child items of the selected items. After clicking this option a second time, click **Refresh** to expand the items in the **Electrical Engineer** tree view.

- You can filter the display of the **Electrical Engineer** and show only the selected branches of the network structure by clicking **Show Only** on the **View** menu.
  To display all the existing data in the **Electrical Engineer**, close the **Electrical Engineer** and open a new session by clicking on the main toolbar.
  Also, you can display the selected branches of the network structure in a new **Electrical Engineer** window by clicking **Show in New Window** on the **View** menu.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Create a Daisy-Chain Connection Pattern*, page 231
- *Drag-and-Drop Rules for the Electrical Engineer*, page 82
- *Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer*, page 83
- *Working with the Electrical Index: An Overview*, page 44

Electrical Engineer Common Tasks

The following tasks are used frequently when you work with the **Electrical Engineer**.

Create Electrical Associations Using the Drag-and-Drop Operation

This procedure explains how to make electrical association among equipment using the drag-and-drop operation. You select an item in the **Electrical Index** and drag it to the appropriate place in the **Electrical Engineer**. For more information, see *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81.

Change Existing Electrical Associations

You change an existing electrical association among equipment by dragging an item in the **Electrical Engineer** and dropping it on another item in the **Electrical Engineer**. For more information, see *Change an Existing Electrical Association*, page 85.

Dissociate Items

This procedure shows you how to dissociate an item from its feeding item in the **Electrical Engineer**. For more information, see *Dissociate an Item from its Feeding Item*, page 90.

Find Other Parent Items in the Electrical Engineer

This option allows you to find all other parent items that are associated with a selected item on a higher hierarchy level in the **Electrical Engineer**. For example, you can select a bus and find all other incomers that are associated with this bus. For more information, see *Find Other Parent Items in the Electrical Engineer*, page 91.

Create a Daisy-Chain Connection Pattern

This topic provides the rules for connecting electrical items in a daisy-chain pattern. For more information, see *Create a Daisy-Chain Connection Pattern*, page 231.

Copy Power Network Structures from Another Plant

You use this feature to copy power network structures from the **Reference Electrical Engineer**. For more information, see *Working with Reference Electrical Engineer: An Overview*, page 44 and *Copy Power Network Structures from Another Plant*, page 93.

Create Electrical Associations Using the Drag-and-Drop Operation

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icon or by double-clicking the parent folder.

2. Highlight the item that you want to associate.

3. In the **Electrical Engineer**, expand the item hierarchy where you want to make the new association. Click the + icon or click the relevant item.

4. Drag the highlighted item from **Electrical Index** to the **Electrical Engineer** and drop it onto the desired target item.

Note

- There are specific rules that govern electrical associations. These rules determine electrical associations of items in the **Electrical Engineer**. For more information, see *Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer*, page 83.

Related Topics

- *Associating Electrical Equipment Common Tasks*, page 215
- *Automatic Creation of Feeder Circuits*, page 218
- *Change an Existing Electrical Association*, page 85
- *Create a Daisy-Chain Connection Pattern*, page 231
- *Dissociate an Item from its Feeding Item*, page 90
- *Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer*, page 83
- *Working with the Electrical Engineer: An Overview*, page 46
- *Working with the Electrical Index: An Overview*, page 44

Drag-and-Drop Rules for the Electrical Engineer

You associate electrical equipment by dragging items in the **Electrical Engineer**. There are specific rules that govern electrical associations. The following rules apply when dragging an item within the **Electrical Engineer**:

Source Item	Target Item	Result
Loads, converting electrical equipments, primary transformer windings, cables, incomer circuits, junction boxes, local panels, miscellaneous electrical equipment	Feeder circuits, cables	The source item becomes the child of the target item in the **Electrical Engineer**.
Bus riser circuits	Coupler circuits	The source item becomes the child of the target item in the **Electrical Engineer**.
Cables, incomer circuits	Generators, battery banks, UPS's, battery chargers, converting electrical equipment, secondary transformer windings	The source item becomes the child of the target item in the **Electrical Engineer**.
Panel - Circuits	Cables	The source item becomes the child of the target item in the **Electrical Engineer**.
Cables	Panel - Circuits	The source item becomes the child of the target item in the **Electrical Engineer**.
Any item that has an electrical parent item (except for buses)	The root of the **Electrical Engineer** tree view	The software dissociates the source item from its parent item.
Cables associated with loads	Loads	The software creates a daisy-chain connection pattern between the loads.

Special Conditions

Changing the association of a terminated power cable by dragging it to another equipment item in the **Electrical Engineer**, disconnects this cable from the terminal strip of that equipment. The software disconnects the cable without displaying a warning message.

If the dragged item has no hierarchical parent item:

- The system creates a new relation with the parent item.

- The system copies all the existing associations from the target item to the dragged item. The copied associations include the highest item in the hierarchy, such as a load or an incomer circuit.

If the dragged item has a hierarchical parent item:

- Dragging it to another item that is not its parent item dissociates the dragged item from its current parent item.

- The dragged item acquires a new electrical relation with the item to which it is dragged. The copied associations include the highest item in the hierarchy, such as a load or an incomer circuit.

- The system copies all the existing associations from the target item to the dragged item. The copied associations include the highest item in the hierarchy, such as a load or an incomer circuit.

- You can connect electrical items in a daisy-chain pattern. For more information, see *Create a Daisy-Chain Connection Pattern*, page 231.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Create a Daisy-Chain Connection Pattern*, page 231
- *Working with the Electrical Engineer: An Overview*, page 46
- *Working with the Electrical Index: An Overview*, page 44

Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer

You can create electrical associations among items by dragging items from the **Electrical Index** to the **Electrical Engineer**. There are specific rules that govern electrical associations. The following rules apply when dragging an item from the **Electrical Index** and dropping it on an item in the **Electrical Engineer**:

Source Item in the Electrical Index	Target Item in the Electrical Engineer	Result	Pre-Conditions
Cables	Generators	The source item becomes the child item of the target generator in the **Electrical Engineer**.	The source item has no parent item in the **Electrical Engineer**.
Motors, heaters, resistors, capacitors, harmonic filters, battery banks, heat traces, UPSs, primary transformer windings, battery chargers, miscellaneous electrical equipment, other converting equipment	Generators	The source item becomes the child item of the target generator in the **Electrical Engineer**.	The source item has a parent cable in the **Electrical Engineer** and that cable has no electrical parent item.
Cables	Battery banks	The source item becomes the child item of the target battery bank in the **Electrical Engineer**.	The source item has no parent item in the **Electrical Engineer**.
Motors, heaters, resistors, harmonic filters, heat traces, UPSs, miscellaneous electrical equipment	Battery banks	The source parent cable becomes the child item of the target battery bank in the **Electrical Engineer**.	The source item has a parent cable in the **Electrical Engineer** and that cable has no electrical parent item.

Source Item in the Electrical Index	Target Item in the Electrical Engineer	Result	Pre-Conditions
Cables	UPSs	The source item becomes the child item of the target UPS in the **Electrical Engineer**.	The source item has no parent item in the **Electrical Engineer**.
Motors, heaters, resistors, harmonic filters, battery banks, heat traces, variable frequency drives (VFDs), miscellaneous electrical equipment	UPSs	The source parent cable becomes the child item of the target UPS in the **Electrical Engineer**.	The source item has a parent cable in the **Electrical Engineer** and that cable has no electrical parent item.
Cables	Capacitors	The source item becomes the child item of the target capacitor in the **Electrical Engineer**.	The source item has no parent item in the **Electrical Engineer**.
Cables	Secondary transformer windings	The source item becomes the child item of the target secondary transformer winding in the **Electrical Engineer**. The dragged item inherits all the target transformer relations.	The source item has no parent item in the **Electrical Engineer**.
Resistors, capacitors, harmonic filters, heat traces, UPSs, battery chargers, VFDs, miscellaneous electrical equipment	Secondary transformer windings	The source item becomes the child item of the target secondary transformer winding in the **Electrical Engineer**. The dragged item inherits all the target transformer relations.	The source item has a parent cable in the **Electrical Engineer** and the target transformer is associated with a circuit.
Cables	Battery chargers	The source item becomes the child item of the target battery charger in the **Electrical Engineer**.	The source item has no parent item in the **Electrical Engineer**.

Source Item in the Electrical Index	Target Item in the Electrical Engineer	Result	Pre-Conditions
Capacitors, harmonic filters, battery bank	Battery chargers	The source parent cable becomes the child item of the target battery charger in the **Electrical Engineer**.	The source item has a parent cable and the parent cable has no parent item in the **Electrical Engineer**.
Cables	Variable frequency drives (VFDs)	The source item becomes the child item of the target VFD in the **Electrical Engineer**. The dragged item inherits all the target VFD relations.	The source item does not have a parent cable in the **Electrical Engineer**.
Motors, heaters, capacitors, harmonic filters, transformer windings, miscellaneous electrical equipment, other converting equipment	VFDs	The source parent cable becomes the child item of the target VFD in the **Electrical Engineer**. The dragged item inherits all the target VFD relations.	The source item has a parent cable and the parent cable has no parent item in the **Electrical Engineer**.
Motors, heaters, resistors, capacitors, harmonic filters, battery banks, heat traces, UPSs, primary transformer windings, battery chargers, VFDs, cables, other converting equipment	Circuits	The source item becomes the child item of the target circuit in the **Electrical Engineer**. The software assigns the dragged item to the parent item of the target circuit: bus, cell, or PDB.	The source item is not associated with the target circuit. The circuit must be a feeder.

Source Item in the Electrical Index	Target Item in the Electrical Engineer	Result	Pre-Conditions
Motors, heaters, resistors, capacitors, harmonic filters, battery banks, battery chargers, heat traces, UPSs, primary transformer windings, VFDs, cables, miscellaneous electrical equipment, other converting equipment	Cables	The source item becomes the child item of the target cable in the **Electrical Engineer**. The dragged item inherits all the target cable relations.	The source item has no parent item and the target cable has no child items in the **Electrical Engineer**.
Panel-circuits	Cables	The source item becomes the child item of the target cable in the **Electrical Engineer**. The dragged item inherits all the target cable relations.	The source panel circuit has no parent item. The target cable has no child items.
Cables	Panel-circuits	The source item becomes the child item of the target panel-circuit in the **Electrical Engineer**. The dragged item inherits all the target panel-circuit relations.	The source cable has no parent items.
Junction box, local panels	Cables	The source item becomes the child item of the target cable in the **Electrical Engineer**. The dragged item inherits all the target cable relations.	The source item has no child items and the target cable has no child items in the **Electrical Engineer**.

Source Item in the Electrical Index	Target Item in the Electrical Engineer	Result	Pre-Conditions
Cables	Junction boxes, local panels	The source cable becomes the child item of the target item in the **Electrical Engineer**. The dragged cable inherits all the target junction box or local panel relations.	The source item has no parent items. The target item has no child items in the **Electrical Engineer**.
Cables (non-parallel power cables)	Miscellaneous electrical equipment	The source cable becomes the child item of the target miscellaneous electrical equipment item in the **Electrical Engineer**. The dragged cable inherits all the target miscellaneous electrical equipment item relations.	

Special Conditions

The software creates the following electrical items automatically in the **Electrical Engineer** when you create them in the **Electrical Index**. These items appear at the root of the **Electrical Engineer** tree view after you create them in the **Electrical Index** even if they do not have an existing electrical relation.

- Buses
- Generators
- Battery banks
- UPSs
- Battery chargers
- Transformers

You can connect electrical items in a daisy-chain pattern. For more information, see *Create a Daisy-Chain Connection Pattern*, page 231.

You cannot dissociate the following items, that is, you cannot drag them to the root of the **Electrical Engineer**:

- Buses

- Feeder circuits

- Coupler circuits

- Transformer secondary winding

Related Topics
- *Working with the Electrical Engineer: An Overview*, page 46
- *Working with the Electrical Index: An Overview*, page 44

Change an Existing Electrical Association

1. In the **Electrical Engineer**, navigate to the item whose association you want to change.

2. Expand the item hierarchy to the location of the new association. Click the + icon or click the items.

   💡 **Tip**

   - To facilitate the drag-and-drop operation and to eliminate the need to scroll up or down in the **Electrical Engineer**, open another **Electrical Engineer** window and navigate to the desired location in the tree view before you start dragging an item.

3. Drag the desired item to the new location in the **Electrical Engineer** and drop it on the item with which you want to associate it.

   📝 **Notes**

   - Whenever you drag a load or any equipment item from one bus to another, the software automatically creates a new feeder circuit. The old feeder circuit from which you dissociated the dragged item remains in its original place. For more information, see *Automatic Creation of Feeder Circuits*, page 218.

   - If you choose to automatically create a feeder circuit with a cell, make sure that in the **Properties** window for the new cell, you select the **In Use** setting for the **Cell Usage** property.

   - If you want to dissociate a load from a circuit and this is the last associated load, make sure that in the **Properties** window for the appropriate cell, you select the **Installed Spare** setting for the **Cell Usage** property.

- There are specific rules that govern electrical associations. These rules determine how you can change an existing electrical association of a particular item. For more information, see *Drag-and-Drop Rules for the Electrical Engineer*, page 82.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Automatic Creation of Feeder Circuits*, page 218
- *Create a Daisy-Chain Connection Pattern*, page 231
- *Drag-and-Drop Rules for the Electrical Engineer*, page 82
- *Working with the Electrical Engineer: An Overview*, page 46

Dissociate an Item from its Feeding Item

1. In the **Electrical Engineer**, navigate to the item that you want to dissociate.

2. Do one of the following:

   - Right-click an item, and on the shortcut menu, click **Dissociate.**

   - Select the item you want to dissociate and drag it to the root of the tree view of the **Electrical Engineer**.

3. At the prompt, click **Yes** to confirm the dissociation.

Notes

- The dissociated item moves to the root of the tree view of the **Electrical Engineer** provided that it is associated with at least one other electrical item. If the dissociated item has no other electrical associations, it disappears from the **Electrical Engineer**. Note that the software also dissociates from the feeding item all the items that are associated with the dissociated item.

- When dissociating the last load from a circuit, select the **Installed Spare** setting for the **Cell Usage** property.

- The software can create undesired and unpredictable associations when you try to dissociate a load that is connected to its feeder through more than one branch (for example, a cable or junction box), or when you move one of the feeding branches to another electrical item. To avoid unwanted associations, you must first dissociate the load. Only then can you dissociate the remaining items in the multiple branches. If you do not dissociate the load first, the software may leave undesired associations in the **Electrical Engineer**.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Drag-and-Drop Rules for the Electrical Engineer*, page 82
- *Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer*, page 83

Find Other Parent Items in the Electrical Engineer

1. In the **Electrical Engineer**, navigate to the desired item.

2. Right-click the item and click **Find Other Parent Items**.

3. On the **Find** dialog box, in the **Results** data window, select an item and then click **Apply**.

   💡 **Tips**

   - The software highlights the parent item in the **Electrical Engineer**.
     You can repeat the procedure to find another parent item.

   - To select a single item and close the **Find** dialog box, click **OK**.

   Related Topics
   - *Associating Electrical Equipment Common Tasks*, page 215
   - *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81
   - *Working with the Electrical Engineer: An Overview*, page 46

Working with the Reference Electrical Engineer: An Overview

You use the **Reference Electrical Engineer** to copy existing partial or complete power network structures by dragging items from one or more specified plants to the **Electrical Engineer** in the current plant.

While working in your project plant, SmartPlant Electrical allows you to connect to another plant and open the **Electrical Engineer** from which you can drag a power network structure to the **Electrical Engineer** in your project. This way, any external plant can serve as a reference for your project plant. Such an external plant can hold numerous assemblies in its **Electrical Engineer**, from which you can drag an assembly to your project **Electrical Engineer**. As you copy an assembly to your project **Electrical Engineer**, the software automatically copies to the **Electrical Index** all the electrical items contained in the assembly as well as the items directly associated with the items displayed in the **Electrical Engineer** (for example, a control station). Furthermore, the software copies the reference power cables and typical schematics to the **Reference Data Explorer**.

The rules that determine where you can drag certain items in the target **Electrical Engineer** are similar to the drag-and-drop rules for items that you drag from the **Electrical Index** and within the **Electrical Engineer**.

Important

- Before connecting to a reference plant and opening its **Electrical Engineer**, make sure that you have been granted full access rights to that plant.

The main features of the **Reference Electrical Engineer** are:

Toolbar

The toolbar contains the following options:

Find - Click **Find** to open the **Find** dialog box where you can look for a specific item in the Electrical Engineer. Click the arrow to display a menu with the following options:

- **Find** - Opens the **Find** dialog box where you can look for a specific item in the **Reference Electrical Engineer**.
- **Close** - Closes the active **Reference Electrical Engineer** window.

Refresh - Click **Refresh** to update the **Reference Electrical Engineer** display. This feature is useful where multiple users are working on the same set of data.

Buttons - Buttons provide a useful shortcut to an item or a group of items. Click **Buttons** ⚙ ▾ to toggle show / hide for existing buttons. To create a button for the currently selected item or folder, click the arrow and click **Add Button**. To remove a button, select the button to remove, then click the arrow, and click **Remove Button**. Note that if you do not select any button to remove, the software removes the uppermost button in the list.

Tree View Pane

The tree view pane of the **Reference Electrical Engineer** shows electrical items organized in the **Reference Plant** folder. All items that appear under this folder have read-only status.

Expand or collapse the hierarchy by clicking the **+** and **-** icons respectively or by clicking an item in the **Reference Plant** folder hierarchy.

📝 Notes

- You can open several instances of the **Reference Electrical Engineer** by clicking **View > New > Reference Electrical Engineer**.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Copy Power Network Structures from Another Plant*, page 93
- *Drag-and-Drop Rules for the Electrical Engineer*, page 82
- *Rules Governing the Copying of Power Network Structures to the Electrical Engineer*, page 95
- *Working with the Electrical Engineer: An Overview*, page 46

Copy Power Network Structures from Another Plant

⊕ Important

- Before connecting to a reference plant and opening its **Electrical Engineer**, make sure that you have been granted full access rights to that plant.

1. Click **Window** > **New** > **Reference Electrical Engineer**.

2. On the **Open Plant Structure** dialog box, select a plant from which you are going to copy the data structure. If the plant that you want does not appear, click **Site Server** to navigate to another location.

3. Click **Open** to display the plant in the **Reference Electrical Engineer**.

4. Expand the folder hierarchy by clicking the + icon beside the folder that you require, or by double-clicking the folder.

5. In the **Electrical Engineer** for the current plant, expand the item hierarchy to display the item onto which you want to copy the network structure.

6. Drag the parent (node) item of the network structure from the **Reference Electrical Engineer** to the **Electrical Engineer** and drop it onto the desired item.

✎ Notes

- You can open more than one **Reference Electrical Engineer** and copy data from different plants to your current plant.

- You can drag a parent item from the **Reference Electrical Engineer** to the topmost root of your current plant.

- You can open an instance of the **Reference Electrical Engineer** for your current plant to duplicate your current plant data. In this case, the results you obtain are not the same as if you move items within the **Electrical Engineer**. For details of how the software copies data from the **Reference Electrical Engineer** to the **Electrical Engineer**, see *Rules Governing the Copying of Power Network Structures to the Electrical Engineer*, page 95.

- When you copy items from the **Reference Electrical Engineer** to the **Electrical Engineer**, the item does not appear in the **Electrical Index**. To update the item in the **Electrical Index**, click **Refresh** 🔃.

Related Topics

- *Associating Electrical Equipment Common Tasks*, page 215
- *Rules Governing the Copying of Power Network Structures to the Electrical Engineer*, page 95
- *Working with the Reference Electrical Engineer: An Overview*, page 92

Rules Governing the Copying of Power Network Structures to the Electrical Engineer

When you create new power network structures in your current plant by dragging items from the **Reference Electrical Engineer** to the **Electrical Engineer**, there are specific rules that govern how the software creates new items and the electrical associations with the items that already exist in the target plant. When you drag an item to the **Electrical Engineer**, the software includes that item and all items that are below it in the hierarchy, which together constitute the scope of copied items.

The following rules apply to dragged items:

- The software assigns names to the items created in the target plant according to the target plant naming conventions.

- Items that the software creates in the target plant inherit the properties of their source items.

- If you drag a load to the root (**Plant**) of the target plant, the software copies the load, but does not associate it to any item in the **Electrical Engineer** - instead the load appears in the appropriate folder of the **Electrical Index**.

- The software copies the electrical relationships that existed between the items in the scope to the target plant. The software creates a new relationship between the dragged item at the top of the hierarchy and the target item. This new relationship follows the rules that apply when dragging items within the **Electrical Engineer**. For more information, see *Drag-and-Drop Rules for the Electrical Engineer*, page 82.

- Cables that are created in the target plant include the conductor arrangement of the source and all the sizing data. The software copies the reference cables to the **Reference Data Explorer** in the target plant and associates them with the project power cables.

- When the scope includes buses, the software copies their PDBs and PDB-bus associations to the target plant.

- When the scope includes circuits that have cells, the software copies these cells and their circuit associations to the target plant.

- When the scope includes circuits that contain internal components, the software copies the circuits and their internal components to the target plant.

- When copying loads or circuits, the software also copies any signal data associated with these items to the target plant.

- When copying an item that is associated with a control station, the software copies the pertinent control stations and control station cables to the **Electrical Index** in the target plant.

- When copying a load that is associated with a typical schematic, the software copies the relevant typical schematics and typical blocks to the **Reference Data Explorer** in the target plant.

Related Topics

Working with the Tabular Editor: An Overview

You use the **Tabular Editor** for viewing and performing editing or other operations on one or more items of a specified item type.

The **Table Properties** dialog box allows you to define the items you want to see in the table view. You customize a filter and layout for the display on the **Advanced Table Properties** dialog box.

The main features of the **Tabular Editor** are:

Header

The header displays the following information:

Total count - The total number of rows (tags) that your table contains. Filtering does not change this value.

Item type - The item type on which your table is based.

Layout - The layout name that your table uses to determine which properties appear, and their order.

Data

You can populate data viewed in the table in one of the following ways:

- Selecting a table cell and typing the appropriate information.
- If the property has a select list associated with it, then you choose the value from the select list.
- If the cell is populated by a value that results in calculating another cell, the software performs the calculation and displays the result in the appropriate cell.

If the cell is populated by a read-only property or a property that is only automatically generated by validation or calculation, you cannot edit that cell. You can select an entire row, corresponding to the properties of a single item tag, by clicking the corresponding row number to the left of the table. Then you can view the properties in the **Properties** window and edit item properties there, too.

The contents of cells can be copied to other cells, provided the cell contents are compatible.

Data Manipulation

You can perform the following operations in the **Tabular Editor**:

- Move a cell from one location to another like in Excel.

- Change the width of a cell by dragging the edge of the cell header and making the cell wider or narrower. When you save the settings, the software retains cell widths the next time you open the layout.

- Freeze one or more columns to prevent them from scrolling. Click a column header to select the column, then right-click in the table and select **Freeze Panes** on the shortcut menu. The software prevents all the columns to the left of the selected column from scrolling horizontally.

- Double-click a cell to select its contents for editing.

- Click a cell once to select the entire row for copying or pasting data.

- For multiple selection using the Tab or Ctrl keys, click the relevant row numbers.

- Click the upper left empty header to select all the items.

Notes

- When navigating in the **Tabular Editor**, the **Properties** window is enabled. When you select a tag in the **Tabular Editor**, the data for the tag appears in the **Properties** window. Editing a property of a tag in the **Properties** window is immediately reflected in the **Tabular Editor** and vice versa.

- The **Properties** window can display data for either the **Tabular Editor** or for one of the windows where you select an item (**Reference Data Explorer**, **Electrical Index**, or **Electrical Engineer**).

- If you select more than one tag in the **Tabular Editor**, the **Properties** window can display common properties when you choose **Select Set** from the list at the top of the window.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics

Viewing and Bulk Editing Data Common Tasks

The following tasks are used frequently when you view and edit data in the Tabular Editor. You select data of a specified item type and specify a layout which determines the data values that the software displays.

You can select a subset of your data in the Tabular Editor to perform the following operations:

Apply Options to a Set of Tags

You can select a set of tags and apply various options such as lookup tables, circuits, and profiles. For more information, see *Apply Options to a Set of Tags*, page 142.

Batch Size Cables

Perform batch cable sizing for a set of tags. For more information, see *Batch Size Cables*, page 186.

Replace Cables

Perform cable replacement for a set of tags. For more information, see *Replace Cables*, page 188.

Associate External Documents with Electrical Items

Associate a reference document with a set of tags. For more information, see *Associate External Documents with Electrical Items*, page 381.

Generate Schematics in Batch Mode

Generate a schematic for a set of loads. For more information, see *Generate Schematics in Batch Mode*, page 287.

Associate Loads with a PDB in Batch Mode

Perform batch load association for a set of tags. For more information, see *Associate Loads with a PDB in Batch Mode*, page 219.

Generate a Report

Generate a report for a set of tags. For more information, see *Generate a Report*, page 387.

Related Topics
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38
- *Using Naming Conventions*, page 127

Display the Tabular Editor

- Click **Window** > **New** >**Table** ▦.

Related Topics
- *Working with the Tabular Editor: An Overview*, page 96

Define a New Tabular Editor View

1. Click **Window > New > Table**.

2. On the **Table Properties** dialog box select an item type from the **Item Type** list.

3. Select a filter from the **Filter** list.

4. Select a layout from the **Layout** list.

📝 **Notes**

- When you create a Tabular Editor view for abstract item types such as loads or converting electrical equipment, as opposed to specific item types such as motors or transformers, the software displays all the common properties for these item types in the Tabular Editor. With loads, for example motors and heaters both appear, but the software cannot display their specific properties in one common Tabular Editor view. However, when you select items of the same specific type, all of the properties appear in the **Properties** window.

- If you do not find a filter you want under **Filter**, you can define a filter on the **Filter** tab of the **Advanced Table Properties** dialog box. Open the **Advanced Table Properties** dialog box by clicking **Advanced** on the **Table Properties** dialog box.

- If you do not find a layout you want under **Layout**, you can define a layout on the **Layout** tab of the **Advanced Table Properties** dialog box. Open the **Advanced Table Properties** dialog box by clicking **Advanced** on the **Table Properties** dialog box.

- To modify an existing table, right-click in the table and select **Edit View**. The **Table Properties** dialog box opens.

Related Topics
- *Working with the Properties Window: An Overview*, page 108

Define a Filter for a Tabular Editor View

1. Click **Advanced** on the **Table Properties** dialog box.

   💡 **Tip**

   - You can open the **Table Properties** dialog box by right-clicking in the Tabular Editor and choosing **Table Properties** from the shortcut menu.

2. On the **Filter** tab of the **Advanced Table Properties** dialog box, specify a name for the filter in **Base filter**.

   💡 **Tip**

   - Select **Default** if you want this filter to be the default for the item type that you selected on the **Table Properties** dialog box.

3. Click **Browse** if you want to search for a filter or create a new filter - either simple or compound - using the **Select Filter** dialog box.

4. To further define a filter, populate the **Definition** box with properties and values. Click **Add** to add a line to the **Definition** box.

5. Select a property, an operator, and a value from the **Edit** group. **Property** lists properties that apply to your item type; **Operator** lists all the possible relationships, such as greater than and not equal to; select the specific property value you want to filter for in **Value**.

   💡 **Tip**

   - You can use an operator with a property that has a value with a unit of measure. For example, you can filter a table showing motors to show only those motors that have the brake power value greater than the value you set in the filter.

6. To add more entries, repeat steps 4 and 5.

7. Click **Delete** if you want to delete the current row in the **Definition** box.

8. Click either **Match all** or **Match any**.

9. Click **OK** on the **Advanced Table Properties** dialog box and again on the **Table Properties** dialog box so that the software can populate your table.

   💡 **Tip**

   - After you have defined a specific filter for your Tabular Editor, you can save that definition. For details, see *Save a Tabular Editor Layout*, page 105.

Define a New Layout for a Tabular Editor View

1. Open the **Table Properties** dialog box by doing one of the following:

   - Click **Window > New > Table Properties**.

   - Right-click in the Tabular Editor and on the shortcut menu, click **Table Properties**.

   💡 **Tip**

   - You can define a layout by selecting a default layout from the **Layout** list and modifying it. You can then save and use the modified layout. For a list of the default layouts that come shipped with the software, see *Shipped Default Layouts*, page 103.

2. Click **Advanced**.

3. On the **Layout** tab of the **Advanced Table Properties** dialog box, specify a name for the layout.

   💡 **Tip**

   - Select **Default** if you want this layout to be the default table layout for the item type, which you specified on the **Table Properties** dialog box. Since there can only be one default layout per item type, whichever layout you last selected as the default becomes the current default layout.

4. Click **Save** if you want to save the layout in order to apply it to another Tabular Editor, click **Delete** if you want to delete the layout.

5. To further define a layout, populate the **Definition** box with display and sort properties.

6. Click **Add** to add a line entry to the **Display Property** list. Then specify the property and caption in the **Edit** group at the bottom of the dialog box.

7. To add and define more display properties, repeat step 5.

   💡 **Tip**

   - To change the display order of the columns in the Tabular Editor, select a row in the **Display Property** box and press **Move Up** or **Move Down**.

8. In the **Sort Property** list click **Add** to add an entry. Then select the property, order, and type in the **Edit** group at the bottom of the dialog box.

9. To add and define more sort properties, repeat step 7.

   💡 **Tip**

   - To change the sort order of the properties in the Tabular Editor, select a row in the **Sort Property** box and press **Move Up** or **Move Down**.

   🔷 **Important**

   - The **Add**, **Insert**, **Delete**, **Move Up**, and **Move Down** buttons apply to the active list view, which can be either **Display Property** or **Sort Property**.

Shipped Default Layouts

Several default layouts come shipped with the software. You can use them to define your own layouts by changing the layout name, adding or deleting properties as required, and saving under a new name.

The following shipped layouts are available for the appropriate item types:

- Default Battery Bank List
- Default Bus List
- Default Cable List
- Default Circuit List
- Default Circuit Breaker List
- Default Control Stations List
- Default Document List
- Default Generator List
- Default Load List
- Motor List
- Default PDB List
- Default Reference Cable List

Related Topics
- *Define a New Layout for a Tabular Editor View*, page 102
- *Layout Tab (Advanced Table Properties Dialog Box)*, page 779

Modify the Table Properties of an Existing Tabular Editor View

1. Click inside the Tabular Editor.

2. Click **View > Table Properties** to open the **Table Properties** dialog box.

   💡 **Tip**

   - You can open the **Table Properties** dialog box by right-clicking in the Tabular Editor and choosing **Table Properties** from the shortcut menu.

3. Review and edit the item type, filter, and layout.

4. If you want to customize a filter or layout click **Advanced** on the **Table Properties** dialog box.

   Define a Filter for a Tabular Editor View, page 101

   Define a New Layout for a Tabular Editor View, page 102

5. Click **OK** on the dialog box to display the modified view.

   📝 **Notes**

   - After defining a layout, you can save it if you want to apply it to another Tabular Editor view. To save the view layout, specify a name in **Name** and click **Save** on the **Advanced Table Common Properties Layout tab.**

   - You can also name and save the entire Tabular Editor view, layout and all:

   Save a Tabular Editor Layout

   - You can edit some of the properties of those items that appear in the Tabular Editor. You can edit the contents of some cells directly in the Tabular Editor if the corresponding property is not read-only or generated strictly by validation; you can copy, cut, and paste properties too, with the same restriction, or you can click on the row number to the left of the Tabular Editor to select an item and edit its properties in the **Properties** window.

   Related Topics
   - *Working with the Tabular Editor: An Overview*, page 96

Save a Tabular Editor Layout

- After you have customized the Tabular Editor, on the **Tabular Editor Property** dialog box, **Layout** tab, click **OK**.

📝 Notes

- You can delete a saved view, too. For more details, see *Delete a Tabular Editor Layout*, page 106.

- The Tabular Editor is where you can display database items.

- You can customize the Tabular Editor. For more details, see *Modify the Table Properties of an Existing Tabular Editor View*, page 104.

Related Topics
- *Customizing Data Display in the Properties Window: An Overview*, page 111
- *Working with the Tabular Editor: An Overview*, page 96

Delete a Tabular Editor Layout

1. Click inside the Tabular Editor.

2. Click **View > Table Properties**.

3. Click **Advanced**.

4. On the **Advanced Table Properties** dialog box, click the **Layout** tab.

5. From the **Name** list, select the layout you want to delete.

6. Under the **Name** group box, click **Delete**.

7. When prompted, click **Yes** to confirm the deletion.

Related Topics
- *Working with the Tabular Editor: An Overview*, page 96

Update Tabular Editor Data

- Right-click inside the Tabular Editor and on the shortcut menu, click **Refresh**.

📝 Note

- The database can change at any time. The software does not update the Tabular Editor automatically: you must update it by using the **Refresh** command manually.

Related Topics
- *Working with the Tabular Editor: An Overview*, page 96

Search for Tags from the Tabular Editor

1. Open a Tabular Editor view.

2. Select a row.

3. Right-click, and on the shortcut menu, click one of the following:

   - **Find in Electrical Engineer**.

   - **Find in Electrical Index**.

   Related Topics
   - *Working with the Tabular Editor: An Overview*, page 96

Use AutoFilter in the Tabular Editor

1. Turn AutoFilter on by right-clicking in the Tabular Editor and choosing **Auto Filter** from the shortcut menu.

   💡 **Tip**

   - Click **AutoFilter** again from the shortcut menu to turn the AutoFilter feature off.

2. Click on the down-arrow at the top of the column you want to filter values for.

   - To match one criterion, choose that criterion from the list that is displayed.

   - To display rows that meet two conditions, choose **Custom** from the list. The **Custom AutoFilter** dialog box opens. Enter the comparison operator and value you want, and then click the **And** button. In the second comparison operator and value boxes, enter the operator and value you want.

   - To display rows that meet either one condition or another condition, enter the comparison operator and value you want, and then click the **Or** button. In the second comparison operator and value boxes, enter the operator and value you want.

   - You can also choose **Blanks** or **NonBlanks** from the list in order to display items that either have no value in the column or do have a value in the column, respectively.

   📝 **Note**

   - Since properties with units of measure are strings, you cannot perform mathematical operations on these type of properties.

Copy Data from the Tabular Editor to an External File

1. Open a Tabular Editor view.

2. Select a row, cell, or group of cells.

   Tip

   - You can click the upper left empty header to select all the items.

3. Right-click inside the Tabular Editor and on the shortcut menu, click **Copy**.

4. Open an external file, for example a Microsoft Word file or an Excel sheet.

5. Paste the data into the desired location.

Note

- In Word, the data is pasted in with tab separators between columns and paragraphs between rows. In Excel, the data is pasted into the same cell positions as the original cell positions in the Tabular Editor.

Related Topics

- *Working with the Tabular Editor: An Overview*, page 96

Working with the Properties Window: An Overview

The **Properties** window display is a two-column table that provides the common properties of the current item tag in the active window (**Reference Data Explorer**, **Electrical Index**, **Electrical Engineer**) or **Tabular Editor**. The active items always determine the content of the **Properties** window. There can only be one **Properties** window open. If only one tag is selected in the active window or in the **Tabular Editor**, or all tags are of the same type, all the properties of that tag type are displayed. If the selected items have different values for their common properties, the value field is blank. For select lists, there is a separating line below which all the available values appear. You can modify the values for these properties in the **Properties** window.

At the top of the **Properties** window is the **Properties** box, listing all of the selected items. Item properties can be displayed by locating the appropriate item in the **Properties** box, as well as by selecting in the **Reference Data Explorer**, **Electrical Index**, **Electrical Engineer**, **Tabular Editor**, or open a single line diagram. If more than one item is selected in the active window, then the list at the top of the **Properties** window contains the choice **Select Set**, and individual items in the set can be selected from the list.

You can use the **Copy Bulk Properties** and **Paste Bulk Properties** buttons to copy properties from one item to another. This action can be particularly useful when, for instance, changes need to apply to more than one item.

Note

- When you pause your pointer over the buttons on the **Properties** window toolbar, a ToolTip appears, giving the name of the button.

- After exiting SmartPlant Electrical, and starting a new session, the software retains the size and position of all the windows that were open in the previous session.

Related Topics
- *Copy Bulk Properties*, page 110
- *Customize Data Display in the Properties Window*, page 112
- *Display Null Values in the Properties Window*, page 113
- *Generate Error Log Files*, page 120
- *Paste Bulk Properties*, page 111
- *Show Brief Properties*, page 113

Turn the Display of the Properties Window On or Off

1. Display the **Properties** window by doing one of the following:

   - Click **View > Display > Property Grid** on the main menu bar.

   - Right-click in an empty area of the main toolbar and then click **Properties** on the shortcut menu.

2. Use the **Properties** window to view and edit item properties.

Related Topics
- *Customizing Data Display in the Properties Window: An Overview*, page 111
- *Working with the Properties Window: An Overview*, page 108

Review and Edit Item Properties

1. Select the item or items for which you want to review the properties.

2. Display the **Properties** window by doing one of the following:

   - Click **View > Display > Property Grid** on the main menu bar.

   - Right-click the empty area of the main toolbar and then click **Properties** on the shortcut menu.

3. Enter or edit information in the cell next to the appropriate property.

   💡 **Tip**

   - If you have selected more than one item, you can perform global replacement of property values by choosing **Select Set** from the list at the top of the **Properties** window and typing data for the property you want to edit. The data value is then populated in all the items of the select set.

4. Enter or edit information in the cell next to the appropriate property.

   💡 **Tips**

   - You can review and edit item properties in the Tabular Editor too. You can edit the contents of some cells directly in the table if the corresponding property is not read-only or generated strictly by a calculation. Or you can click any cell or on the row number to the left of the table to select an item and edit its properties in the **Properties** window.

   - In place of values that include automatically assigned units of measure, you can type any string preceded by a single quote. This is useful if you do not initially know a specific value and want to insert a comment instead.

Copy Bulk Properties

1. In the **Electrical Index** or Tabular Editor window, select the item for which you want to copy the bulk properties.

2. In the **Properties** window, click **Copy Bulk Properties** .

3. These properties are now available to paste onto another item. For more details, see *Paste Bulk Properties*, page 111.

🖉 Note

- You must have a bulk layout defined for the item type of the selected item.

Related Topics
- *Working with the Properties Window: An Overview*, page 108

Paste Bulk Properties

1. In the **Electrical Index** or Tabular Editor window, select the item or items that you want to paste bulk properties onto.

💡 Tip

- You must first have copied the bulk properties of a different item in order to paste them onto your current selection. For more details, see *Copy Bulk Properties*, page 110.

2. If you have selected to paste onto more than one item, from the tag list at the top of the **Properties** window, choose **Select Set**.

3. Click **Paste Bulk Properties** on the **Properties** window toolbar to paste properties onto your current selection.

Related Topics
- *Working with the Properties Window: An Overview*, page 108

Customizing Data Display in the Properties Window: An Overview

You can customize the display of the **Properties** window by using the following **Properties** window toolbar buttons:

- **Alphabetic** ![icon] - Lists properties in alphabetical order.

- **Categorized** ![icon] - Displays properties grouped by specific categories.

- **Copy Bulk Properties** ![icon] - Copies the bulk properties of a selected item. The properties are then available on the **Paste Bulk Items** button.

- **Display Null** ![icon] - Displays a null value in blank properties that currently contain no value. This option help avoid confusion between empty cells and those containing a zero.

- **Paste Bulk Properties** ![icon] - Pastes copied bulk properties onto the newly selected item or items. You must have previously selected an item or items and copied bulk properties.

- **Show Brief Properties** ![icon] - Displays those properties that have been previously specified as **Brief**.

Related Topics

Customize Data Display in the Properties Window

1. If the **Properties** window is not already displayed, do one of the following:

    *   Click **View > Display > Property Grid** on the main menu bar.

    *   Right-click in an empty area of the main toolbar and then click **Properties** on the shortcut menu.

2. You can display properties alphabetically or by categories by clicking the corresponding display mode button. Click ⬛ to display alphabetically. Click ⬛ to display by categories.

3. You can display only brief properties by clicking **Show Brief Properties** ⬛.

4. You can display a null character in empty properties cells by clicking **Display Null** ⬛.

📝 **Notes**

*   When you point to a button on the **Properties** window toolbar and pause over it, a ToolTip displays the corresponding command.

*   You create categories and assign properties to them in the Data Dictionary Manager.

Related Topics
*   *Working with the Properties Window: An Overview*, page 108

Show Brief Properties

*   Click the **Show Brief Properties** button ⬛ in the **Properties** window.

Display Null Values in the Properties Window

1. Select an item in your drawing.

2. Right-click and select **Properties**.

3. Click the **Display Null** button ⬛ to display a null value in blank properties.

Related Topics
*   *Customizing Data Display in the Properties Window: An Overview*, page 111

Customizing Preferences: An Overview

You can set various preferences for SmartPlant Electrical. The software allows you to set general preferences as well as preferences for SLDs, load summary reports, and load deletions.

The general preferences let you specify whether to open a dialog box after creating or duplicating an item, activating the use of log files for reporting errors after performing some batch operations, displaying in the **Electrical Engineer** buses, incomers, and bus risers together with the associated PDB item tags.

SLD preferences allow you to customize various generation settings such as displaying NULL in the generated drawing when there is no data, setting various margin settings, setting a distance for SLD internal elements, and so forth.

The report settings let you specify the data that the software will use to generate your load summary reports.

The delete load settings allows you to customize which related (sub) items of a main item are deleted when deleting a main item from the **Electrical Index**.

Your preferences customization does not affect the preferences set by other users in the plant. However, a user with Administrator rights can set new defaults and override your user-defined preferences.

Related Topics
- *Preferences Common Tasks*, page 114

Preferences Common Tasks

The following tasks are used frequently when you customize default preferences in SmartPlant Electrical.

Customize General Preferences

General settings allow you to customize default preferences for creating and duplicating new items, displaying PDB item tags in the **Electrical Engineer**, and activating log file generation. For more information, see *Customize General Preferences*, page 115.

Customize Preferences for SLD Generation

SLD preferences allow you to customize default preferences for single line (SLD) generation. For more information, see *Customize Preferences for SLD Generation*, page 115.

Customize All Feeder Load Summary Report Preferences

These options allow you to specify the PDB loads that will account for the coincidence factors and whether and how to account for any loads connected to redundant buses. For more information, see *Customize All Feeder Load Summary Report Preferences*, page 116.

Customize Delete Loads Preferences

This procedure allows you to customize the Delete Loads default preferences that were set in the Options Manager. For more information, see *Customize Delete Loads Preferences,* page 119.

Customize General Preferences

1. Click **File** > **Preferences**.
2. On the **General** tab, under **Open after creating or duplicating an item**, do one of the following:

   - Click **None** to display a created or duplicated item in the **Electrical Index** or the **Reference Data Explorer** without opening either the **Item Tag** or **Common Properties** dialog box, with a default property value and naming convention defined in Options Manager.

   - Click **Item Tag dialog box** to be able to type text for the item tag right after creating or duplicating an item.

   - Click **Common Properties dialog box** to be able to enter the desired electrical properties right after creating or duplicating an item.

3. Under **Electrical Engineer display options for buses, incomers and bus risers**, do one of the following:

   - Select the **Include associated PDB item tags** check box to display in the **Electrical Engineer** item tags of buses, bus risers, and incomers together with the item tags of the associated power distribution boards. For example, MCC \ Bus A.

   - Clear the **Include associated PDB item tags** check box to display in the **Electrical Engineer** item tags of buses, bus risers, and incomers without the associated PDB item tags. For example, Bus A.

4. Under **Log file options**, select **Activate log file generation** to enable generation of various error log files.

5. Click **OK** or another tab.

Related Topics

Customize Preferences for SLD Generation

1. Click **File** > **Preferences**.

2. Click the **SLD** tab.

3. Under **SLD drawings**, do one of the following:

   - Select the **Save settings of last generated drawing** check box to apply settings defined for the last generated drawing to all subsequent generations.

   - Clear the **Save settings of the last generated drawing** check box to save only the SLD margin preferences and return all the other SLD options to the application defaults.

     💡 **Tip**

     - Selecting the **Save settings of the last generated drawing** check box instructs the software to apply all the settings defined on the **Single Line Diagram Options** dialog box apart from the settings defined on the **Preferences** dialog box (**SLD margins** and **Distance for SLD internals**). This preference option takes effect after generating an SLD and applies to all subsequent SLD generations.

4. Do one of the following:

    - Select **Use bus tie symbols** to show a horizontal representation between two buses using the BusTie.sym file. In this case, the software shows the connection between the two buses as a straight bus-tie connection without showing the items connected between and within the circuits. Note that the BusTie symbol is a "circuit" type symbol and has its first connection point on the rightmost side and the second connection point at the leftmost side. Also, the item property in the SLD displays only the coupler information. Furthermore, if you use the **Find in SLD** command, the software finds the pertinent coupler and not the bus riser.

    - Clear the **Use bus tie symbols** check box to use a vertical representation of a connection between two buses. This is the standard bus coupler-riser circuit connection representation that includes both coupler and the riser circuits and displays them as vertical objects, connected to each other. The software displays both circuits as standard arrows.

5. Do one of the following to set your preference for the display of electrical item positions when opening an existing SLD:

    - Select this option to show the position of electrical items as they were in the last saved drawing.

    - Clear this option to revert to the optimized positioning of electrical items by regenerating the SLD.

6. Under **SLD margins**, select the desired unit of measure (mm or inches) and define the margin settings.

7. Under **Distance for SLD internals**, enter values and select a unit of measure for representation of buses, circuits and circuit components in a generated SLD:

    - In **Distance between circuits**, set the horizontal distance between the circuits above and below the buses.

    - In **Distance between buses**, set the horizontal distance between the buses.

    - In **Distance between circuit internals**, set the vertical distance between the circuit internal components (fuses, starters, and so forth).

8. Click **OK** or another tab.

Related Topics

Customize All Feeder Load Summary Report Preferences

1. Click **File** > **Preferences**.

2. Click the **Reports** tab.

3. Under **Calculation method for All Feeder Load Summary Report**, select one of the following:

   - **Use each load coincidence factor** - select to use the coincidence factor of each individual load to calculate the total connected loads on the bus.

   - **Use Bus PDB coincidence factor** - select to use the average coincidence factors of the bus to calculate the total connected loads on the bus.

4. Under **Inclusion of coupled buses in All Feeder Load Summary Report**, select one of the following:

   - **Do not include coupled buses** - select to include in the calculation only those loads that are connected to the current bus and does not include the loads that are connected on the redundant buses.

   - **Include coupled buses connected by couplers only** - select to include in the calculation the loads connected on the current bus as well as loads connected on all the redundant buses that are connected to the current bus by a coupler circuit.

   - **Include coupled buses connected by couplers and bus risers** - select to include the loads of the current bus as well as the loads on all the redundant buses that are connected to the current bus by coupler and riser circuits.

5. Select the **Account for converting equipment power losses** check box to instruct the software to account for power losses when using converting equipment.

   💡 **Tip**

   - The software adds these power losses to the upstream feeding bus and accounts for them as continuous loads. The same losses are also accounted for in the All Feeder Load Summary and PDB Load Summary reports.

6. Click **OK** or another tab.

Related Topics

Customize Delete Loads Preferences

1. Click **File** > **Preferences**.

2. On the **Delete Loads** tab, under **Delete Options** select the options you want to apply when deleting a main item.

3. Click **OK** or another tab.

 Note

- Customizing the **Delete Loads Preferences** in SmartPlant Electrical, overrides the **Delete Loads Preferences** set in Options Manager for the specific user. It does not change Option Manager's settings for other users.

Related Topics

Customize Delete Load Preferences for a Specific Item

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Select the item you want to delete and do one of the following:

   - Right-click and on the shortcut menu select, **Delete**.

   - Click ✖.

   - Click, **Edit** > **Delete**

3. On the **Confirm Item Delete** prompt, click **Delete Options**.

4. Select or clear the options you want to apply when deleting the selected item.

5. Click **Yes**.

 Note

- This procedure overrides the **Delete Load Preferences** for this specific item only. It does not change the **Delete Load Preferences** in the software.

Generating Error Log Files: An Overview

There are several batch operations that you can perform in SmartPlant Electrical, some of which may not be completed or performed due to failures in validations, missing data, and so forth. You can view a list of errors for these operations in an appropriate log file. Among the operations for which the software generates log files are applying profiles, batch cable sizing, replacing cables, batch data deletion, copying and pasting multiple items in the Tabular Editor, and bulk copying and pasting of values in the **Properties** window.

Activate Log File Generation

1. Click **File** > **Preferences**.

2. On the **Preferences** dialog box, under **Log file options** in the **General** tab folder, select **Activate log file generation**.

3. Click **OK**.

Generate Error Log Files

1. Click **File > Open > Log Files**.

2. On the **Log Files** dialog box, select a log file category from the **Log file type** list.

3. Select one of the following:

   - **All log file data** - select to display data and rule inconsistency errors.

   - **Rule inconsistencies only** - select to display rule inconsistency errors only.

   - **All log file data excluding rule inconsistencies** - select to display data errors only.

4. Select **Show log file data for last operation only** to display errors for the last performed operation relating to the specified log file. Clear to display accumulated errors for successive operations.

5. Click **Open**.

   💡 **Tip**

   - The software generates the log file in a Microsoft Excel workbook where you can view a list of any failed operations in the selected category. You can then go back to the **Log Files** dialog box, select another log file type, and click **Open**. The software displays the log file in the previously opened Excel workbook but in another worksheet.

6. Click **Close**.

Note

- The software creates log files only after you activate the log file option on the **General** tab of the **Preferences** dialog box.

Related Topics
- *Activate Log File Generation*, page 120
- *Customize General Preferences*, page 115

Troubleshooting Rule Inconsistencies: An Overview

When you modify the value of any property of an item for which rules are defined in the **Rule Manager** and the software detects a value that is inconsistent with one or more rules, the software responds in accordance with the action specified for the inconsistency. You can view inconsistencies for selected items at the time of data updating, or you can display a summary of rule inconsistency statistics and view a log file with details of the inconsistencies at any time. The software displays inconsistencies for values of item properties and of relationship properties between a main item and an item associated with it.

When viewing inconsistencies at the time of updating, you can choose to accept or reject inconsistencies where the action specified is **Warn**. If you accept the inconsistency, the software saves the data and warns you that the inconsistency exists each time you update the data. If you reject the inconsistency, the software does not update the database with values that violate the rules.

When the specified action is **Reject**, the software does not update the database until you fix the values that violate the rule.

After viewing inconsistencies, you can change the data values manually as needed to meet the rule criteria.

Related Topics
- *Activate Log File Generation*, page 120
- *Display Rule Inconsistency Statistics*, page 122
- *Troubleshoot Rule Inconsistencies*, page 123

Display Rule Inconsistency Statistics

This option allows you to view a summary of rule violations that may have occurred for the selected items. You can also display the log file in Excel format if you require.

1. Click **View** > **Rule Inconsistency Statistics**.

2. On the **Rule Inconsistency Statistics** dialog box, click **Show Log**.

Note

- To make this feature available in SmartPlant Electrical, in the **General Settings** of **Options Manager**, select **Enable** for the **Consistency Rules** setting.

Related Topics
- *Activate Log File Generation*, page 120
- *Customize General Preferences*, page 115
- *Troubleshoot Rule Inconsistencies*, page 123
- *Troubleshooting Rule Inconsistencies: An Overview*, page 120

Troubleshoot Rule Inconsistencies

This procedure applies when you update any data value for an item where there are one or more rule violations. After you click **OK** or **Apply** on a **Common Properties** dialog box, or you update a value in the **Properties** window or the Tabular Editor, you can view the rule inconsistencies.

1. If a message appears prompting you to view rule inconsistencies, click **Yes**.

2. On the **Rule Inconsistencies** dialog box, do one of the following:

   - Click **Reject Operation** to close the dialog box without saving updated data values.

   - Click **Accept Inconsistencies** to close the dialog box and save the data even though the data values violate the rules.

   - Click **Close** to close the dialog box when one or more rule inconsistencies exist for which the action is specified as **Reject**.

   📝 **Note**

   - The commands **Reject Operation** and **Accept Inconsistencies** apply only to inconsistencies for which the action is **Warn**. These commands only appear when there are no inconsistencies that specify the **Reject** action.

Related Topics
   - *Activate Log File Generation*, page 120
   - *Display Rule Inconsistency Statistics*, page 122
   - *Troubleshooting Rule Inconsistencies: An Overview*, page 120

Operator Descriptions and Rule Examples

Describes the operators available on the **Consistency** tab of the **Rule Properties** dialog box. Also, provides examples of rules that you can create using a particular operator.

Note

- In the examples shown in the table, the rule components are displayed in brackets to separate them visually from properties or values. When a string that follows an operator does not start with a bracket, this indicates a single component rule.

Operator	Description	Rule Example
=	functions as 'equal to' with numeric characters or as 'like' with text strings	In a filtered subset of 1hp motors: (Motor) Rated voltage = 120 v
>	greater than	(Cable) Insulation rated voltage > (Motor) Rated voltage
>=	greater than or equal to	(Motor) Rated power >= (Motor) Brake power
<	smaller than	(Motor) Asynchronous speed < (Motor) Synchronous speed
<=	smaller than or equal to	(Motor) Asynchronous speed <= (Motor) Synchronous speed
<>	not equal to	In a filtered subset of heaters with heater rated power above 10kW: (Heater) Number of phases <> 1
IN	any of the specified values	In a filtered subset of 10hp motors: (Motor) Rated voltage In 380 V; 415 V; 460 V Comment: The values are case-sensitive and must be typed exactly as they appear in the Data Dictionary. The value separator must be either a semi-colon or a comma.

Operator	Description	Rule Example
Is Null	ensures whether the user enters values for the item tag belonging to the rule component	Supply ACDC Is Null
Is Not Null	used for testing or comparing values or properties in a conjunction with another rule consistency	(Cable) Design length >= (Cable) Estimated length (Cable) Design length Is not Null (Cable) Estimated length Is not Null Comment: if you do use the Is Not Null operator in this rule, the software treats the expression as true even if the values for the (Cable) Design length and (Cable) Estimated length properties are not defined at all.

Related Topics

- *Rule Components Tab (Rule Properties Dialog Box)*, page 795

Creating Electrical Equipment Common Tasks

The following tasks are used frequently when you create electrical equipment in the **Electrical Index**. These items consist of *loads* such as motors and heaters, and other equipment that includes junction boxes, control stations, and cables.

Create a Project Item

You can create a new electrical item directly in the **Electrical Index**. For more information, see *Create a Project Item*, page 53.

Copy a Reference Item to the Electrical Index

When you copy a reference item containing default data that you use frequently, you save time by using the reference item as a template. In this way, you do not have to type common values every time that you create a new item in the **Electrical Index**. For more information, see *Copy a Reference Item to the Electrical Index*, page 139.

Duplicate an Item

You can duplicate any item in the **Reference Data Explorer** or the **Electrical Index**. This method is particularly useful when you want to create a hierarchy structure that is identical to one that already exists. For more information, see *Duplicate an Item*, page 38.

Apply a Profile

You can apply a *profile* to any item or group of items, and the software creates and associates all the referenced items in the **Electrical Index** with the default data. In this way, you can quickly define common properties for a group of items of the same type, and you can associate the same type of typical circuit, control stations, cables, and typical schematic with each item in the group. For more information, see *Apply a Profile to an Item*, page 147.

Apply Lookup Table Data

You can apply data from a lookup table to any type of equipment for which you have created a lookup table. Each lookup table is specific for an item type, such as transformers, and by selecting a key value for a lookup table, the software automatically populates the item with other properties that you defined in the lookup table. For more information, see *Apply Lookup Table Data to Equipment*, page 143.

Create a Custom Folder

This procedure allows you to create custom folders inside item type folders. Custom folders are used when you want to divide an item type into smaller specific groups according to parameters set by you. For example you could divide power cables into groups according to there power rating. For more information about creating a custom folder, see *Create a Custom Folder*, page 354.

Using Naming Conventions

You define naming conventions per item type, such as motors, cables, static loads, and so forth, in the Options Manager. For full details of how to define naming conventions, see the Options Manager User's Guide: *Naming Conventions: An Overview*.

You set whether to use naming conventions by default for all item types in the Data Dictionary by setting the **Comply with Naming Convention** flag to **True** or **False**.

When set to **True**, each time you create a new project item in the **Electrical Index**, the software requires you to use an item tag that complies with the defined naming convention for the item type. The software prompts you with a new item tag based on the naming convention whenever you create a new item by a **New** command, a **Duplicate** command, by dragging items from the **Reference Data Explorer**, or by applying profiles.

When set to **False**, the software prompts you with a new item tag based on the naming convention, but you can overwrite it with any characters that you want.

Notes

- You can override the default naming convention mode for one or more individual tags by selecting the desired tags, and in the **Properties** window, changing the value of the **Comply with Naming Convention** property.

- If you change the value of the **Comply with Naming Convention** property for an item tag from **False** to **True**, the software designates a new item tag for the item based on the naming convention.

- Whether or not you use the naming convention, the software always prompts you with a new item tag that is unique.

Related Topics

Create an Electrical Motor

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Motors**.

2. Right-click the **Motors** folder, and then click **New Motor** or **New Motor with Heater**.

   💡 **Tip**

   - If you create a motor without a heater, you can add a heater later by right-clicking the motor, and on the shortcut menu, clicking **New Heater**.

3. Accept the default item tag that is based on the *naming convention* for the motor, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Electrical Motor Common Properties** dialog box, edit the motor properties as required. For more details, see *Edit Electrical Motor Common Properties*, page 129.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new electrical motor.

- The **Electrical Motor Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new electrical motor.

- You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

- When you add a motor with a space heater, the heater appears as a sub-item of the motor. The motor symbol includes an indication that it has a space heater 🔵. If you subsequently delete the motor, the software deletes the heater also.

Related Topics
- *Create Load Items in Batch Mode*, page 131
- *Duplicate an Item*, page 38
- *Edit Electrical Motor Common Properties*, page 129
- *Electrical Motor Common Properties Dialog Box*, page 690
- *Working with SmartPlant Electrical Common Tasks*, page 34

Edit Electrical Motor Common Properties

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Load** > **Motors**.

   💡 **Tips**

   - For a motor without a heater, you can add a heater by right-clicking the motor, and on the shortcut menu, clicking **New Heater**.

   - You can delete a heater from a motor by selecting the heater, and on the shortcut menu, clicking **Delete**.

2. Right-click the desired motor and click **Common Properties** to open the **Electrical Motor Common Properties** dialog box.

3. Edit the data as needed on each tab. Note that an asterisk beside a property name indicates required data.

   📝 **Note**

   - For full details of the most commonly used properties, see the **Electrical Motor Common Properties** dialog box. Also, you can edit these properties and additional properties on the **Properties** window or on the Tabular Editor.

Related Topics
- *Create an Electrical Motor*, page 128
- *Duplicate an Electrical Motor*, page 130

Calculate Power Requirements for an Electrical Motor

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Motors**.

2. Do one of the following:

   - Expand the **Motors** folder and select an existing motor from the list.

   - Right-click the **Motors** folder, then create a new motor.

3. Right-click the motor and click **Common Properties** to open the **Electrical Motor Common Properties** dialog box.

4. Click the **Load Data** tab and enter the appropriate load data properties.

5. Click the **Electrical Data** tab to view the calculated power requirements for the motor.

📝 **Notes**

- For full details of the properties that the software uses in power calculations and the formulas the software uses to calculate power requirements, see *Electrical Calculations*, page 531.

- For full details of the most commonly used properties, see the **Electrical Motor Common Properties** dialog box. Also, you can edit these properties and additional properties on the **Properties** window or on the Tabular Editor.

Related Topics
- *Electrical Motor Common Properties Dialog Box*, page 690

Duplicate an Electrical Motor

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Load** > **Motors**.

2. Right-click a motor and click **Duplicate** to make a copy of the motor at the lowest position in the folder.

3. Accept the default item tag that is based on the *naming convention* for the motor, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. If required, on the **Electrical Motor Common Properties** dialog box, edit the motor properties as required. For more details, see *Edit Electrical Motor Common Properties*, page 129.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you duplicate an electrical motor.

- The **Electrical Motor Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you duplicate an electrical motor.

- When you duplicate a motor, the software copies only the item properties, except for the **Item tag** property, which follows the motor *naming convention*. The software does not copy associated items such as cables, typical circuits, or control stations, nor does it duplicate associations to feeder circuits.

- When the motor has a space heater, the software duplicates the space heater as well.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply Lookup Table Data to Equipment*, page 143
- *Create an Electrical Motor*, page 128
- *Creating Electrical Equipment Common Tasks*, page 75
- *Electrical Motor Common Properties Dialog Box*, page 690

Create Load Items in Batch Mode

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads**.

2. In the **Electrical Index**, expand the **Electrical Equipment** folder hierarchy by clicking the + icons or by double-clicking the folders.

3. With the desired folder selected, right-click, and then click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Equipment Profile** tab.

5. Select **Enable equipment profile**.

6. In the **Number of items to create** box, enter the number of new electrical items that you want to create.

7. Beside **Create item based on profile**, click **Find**.

8. On the **Find** dialog box, click **Find Now** to display the available profiles.

9. From the list of profiles, highlight the profile that you want to apply to the newly created items and click **OK**.

10. Click **Apply**.

Notes

- If the **Create item based on profile** box contains the text **No default profile exists**, the software does not create any items when you click **Apply**. You must select a profile first.

- If the profile contains a reference item, the software populates new items that you create by this method with values copied from the reference item.

Create Static Electrical Equipment

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Static Electrical Equipment**.

2. Right-click the folder for the desired item type - for example, **Heaters** - and then click **New**.

3. Accept the default item tag or type your own item tag for the item.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the item type. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Static Load Common Properties** dialog box, edit the item properties as required. Note that an asterisk beside a property name indicates required data.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new item.

- The **Static Load Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new item.

- You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

Related Topics
- *Copy a Reference Item to the Electrical Index*, page 139
- *Duplicate an Item*, page 38
- *Static Load Common Properties Dialog Box*, page 736
- *Working with SmartPlant Electrical Common Tasks*, page 34

Create Miscellaneous Electrical Equipment

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Miscellaneous Electrical Equipment**.

2. Right-click the folder for the desired item type - for example, **Lighting Fixtures** - and then click **New**.

3. Accept the default item tag or type your own item tag for the item.

   💡 **Tips**

   • In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   • In the **Electrical Index**, the name must be in accordance with the *naming convention* for the item type. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Other Electrical Equipment Common Properties** dialog box, edit the item properties as desired. Note that an asterisk beside a property name indicates required data.

   📝 **Notes**

   • On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new item.

   • The **Other Electrical Equipment Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new item.

   • You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

   Related Topics
   • *Copy a Reference Item to the Electrical Index*, page 139
   • *Duplicate an Item*, page 38
   • *Other Electrical Equipment Common Properties Dialog Box*, page 724
   • *Working with SmartPlant Electrical Common Tasks*, page 34

Create a Generator

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Generators**.

2. Right-click the **Generators** folder, and then click **New**.

3. Accept the default item tag or type your own item tag for the generator.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the generator. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the generator properties in the **Properties** window as required.

   📝 **Note**

   - On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create the generator.

   Related Topics
   - *Copy a Reference Item to the Electrical Index*, page 139
   - *Duplicate an Item*, page 38
   - *Working with SmartPlant Electrical Common Tasks*, page 34

Create a Battery Bank

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Battery Banks**.

2. Right-click the **Battery Banks** folder, and then click **New**.

3. Accept the default item tag or type your own item tag for the battery bank.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the battery bank. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the battery bank properties in the **Properties** window as required.

📝 **Note**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create the battery bank.

Related Topics
- *Copy a Reference Item to the Electrical Index*, page 139
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38
- *Rename an Item*, page 39

Create Converting Equipment

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Converting Equipment**.

2. Right-click the folder for the desired item type - for example, **Battery Chargers** - and then click **New**.

3. Accept the default item tag or type your own item tag for the item.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the item type. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Converting Equipment Common Properties** dialog box, edit the item properties as required. Note that an asterisk beside a property name indicates required data.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new item.

- The **Converting Equipment Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new item.

- You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

Create a Transformer

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Converting Equipment** > **Transformers**.

2. Right-click the **Transformers** folder, and then click **New 2-winding Transformer** or **New 3-winding Transformer**.

3. Accept the default item tag or type your own item tag for the transformer.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the transformer. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Converting Equipment Common Properties** dialog box, edit the transformer properties as required. Note that an asterisk beside a property name indicates required data.

   📝 **Notes**

   - On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new transformer.

   - The **Converting Equipment Common Properties** dialog box shows the most commonly used properties. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new transformer.

   - You can also edit the most commonly used properties and additional properties in the **Properties** window or on the Tabular Editor.

   - The secondary windings of a transformer are sub-items of the transformer. The properties of the secondary windings appear separately in the **Properties** window. To view these properties, expand the icon that represents the transformer 🔲 and select the icon representing the desired secondary winding 🔲.

Related Topics
   - *Converting Equipment Common Properties Dialog Box*, page 678
   - *Copy a Reference Item to the Electrical Index*, page 139
   - *Define Transformer Connections*, page 137
   - *Define Transformer Tapping*, page 138
   - *Duplicate an Item*, page 38
   - *Working with SmartPlant Electrical Common Tasks*, page 34

Define Transformer Connections

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Converting Equipment** > **Transformers**.

2. Right-click a transformer primary winding, and on the shortcut menu, click **Connections and Tapping**.

3. On the **Transformer Connections and Tapping** dialog box, click the tab for the winding where you want to make the definitions: **Primary**, **Secondary 1**, or **Secondary 2**.

4. Specify the connection, for example, Wye, Delta, Star.

5. Specify the type of neutral grounding that you require.

   Tip

   - The software does not validate the type of connection that you selected and allows you to specify neutral grounding for all connection types.

6. Where the neutral grounding value is **Solid Grounding** or **Impedance** click the ellipsis button ⬚ beside **Grounding cable** to open the **Find** dialog box, where you can select a grounding cable.

   Tip

   - If necessary, define a new grounding cable, ensuring that you select the **Use for neutral grounding** check box - only those grounding cables for which this check box is selected are available for selection.

7. Where the neutral grounding value is **Impedance** click the ellipsis button ⬚ beside **Grounding resistor**.

   Tips

   - On the **Find** dialog box, the software retrieves all the existing resistors regardless of whether they have or have not been associated with transformers. Make sure that you select an appropriate resistor that has not been associated yet.

   - If necessary, define a new resistor.

8. Click **Apply** to accept the values.

9. For a **Secondary** tab, enter values for the connection group, feeder winding, and phase shifts as you require.

Notes

- You must specify a separate grounding cable and resistor for each winding, if needed.

- To enable the display of grounding cable and resistor data in an SLD, you must use a transformer symbol that includes macros for these items. To do this, right-click on the transformer, and on the shortcut menu click **Associate custom symbols**. On the **Associate Custom Symbols** dialog box, under **Document Type,** select **Single Line Diagrams,** then navigate to the symbol folder and select the file **Transformer_Cable_Resistor.sym**.

Related Topics
- *Working with SmartPlant Electrical Common Tasks*, page 34

Define Transformer Tapping

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Converting Equipment** > **Transformers**.

2. Right-click a transformer primary winding, and on the shortcut menu, click **Connections and Tapping**.

3. On the **Transformer Connections and Tapping** dialog box, click the tab for the winding where you want to make the definitions: **Primary**, **Secondary 1**, or **Secondary 2**.

4. Select **Tapping required** and enter values for the tapping, percentage per step, and number of steps as you require.

5. Enter values for other transformer windings as you require by clicking the appropriate tabs.

Related Topics
- *Create a Transformer*, page 136
- *Working with SmartPlant Electrical Common Tasks*, page 34

Copy a Reference Item to the Electrical Index

1. In the **Reference Data Explorer**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders until you can see the item that you want to copy.

2. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders and navigate to the folder of the item type corresponding to the item that you want to copy. For example, to copy a motor, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Motors** and select the **Motors** folder.

3. Drag the item from the **Reference Data Explorer** to the appropriate folder in the **Electrical Index**.

   💡 **Tip**

   - You can copy an item from the **Reference Data Explorer** only to a folder in the **Electrical Index** that can contain items of the same type.

4. Accept the default item tag that is based on the *naming convention* for the item type, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

5. For the item that you copied to the **Electrical Index**, edit the item properties in the **Properties** window.

   📝 **Notes**

   - On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create the item.

   - Some items have a **Common Properties** dialog box from which you can edit the most common properties for the item. You access this dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the item. For full details of the properties, see the appropriate dialog box topic.

   - When you copy a reference item, the software copies the item properties, except for the **Item tag** property, which follows the item *naming convention*.

   - For an item that includes sub-items, such as a typical circuit with internal components, the software copies the sub-items as well. In this case, the sub-items are the circuit components.

Populating Items with Data: An Overview

You can create items and populate them with data in a batch operation, thus keeping to a minimum the editing that you need to perform for individual items. You can perform batch item creation using the following methods:

Applying a typical item to a project item.

Typical items in the **Reference Data Explorer** enable you to create project items based on the selected typical items. The software simply clones the typical item in the **Electrical Index** after dragging a typical item to an appropriate folder in the **Electrical Index**.

Applying options using typical items.

You can use the **Apply Options** feature to create and associate project items with electrical equipment in the **Electrical Index** where the associated items are based on typical items defined in the **Reference Data Explorer**. This option is very useful for populating data in batch mode. For example, you can select a number of motors in the **Electrical Index** and associate them with a control station, a typical circuit, or both. You can also associate these motors with a typical schematic.

Applying a profile to an electrical item.

You can define specific profiles for various item types. A profile is an assembly of various typical items that you can use as a template for creating project items. For example, you can create a motor profile that consists of the following reference items: a motor, one or more cables, a control station, a circuit with protection devices, and schematic. Applying a profile to an electrical item in the **Electrical Index** enables you to specify various properties for that item as they were defined in that profile. This way you can create an electrical item with all its related equipment and with all the properties as you require.

Applying a lookup table to an electrical item.

Lookup tables are data tables used to populate project items singly or in batch with data values based on one or more key fields. Where the key fields exactly match the data for a particular item, the software copies the data from the matching row in the lookup table to the project item. For example, you can select a number of motors in the **Electrical Index** and apply a selected lookup table for a motor with a particular motor rated power and rated voltage. The software will then populate the properties of the selected tables with appropriate values.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Set of Signals to an Item*, page 148
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply a Typical Schematic to an Item*, page 146

Applying Options Common Tasks

The following tasks are used frequently when you use the **Apply Options** command to apply a typical item to a project item in the **Electrical Engineer**.

Apply a Typical Circuit to an Item

Applying a typical circuit to an electrical item in your project allows to create a new circuit with predefined properties.   The software automatically associates the new circuit with the selected project item. You can apply a typical circuit to all loads, all converting equipment items, and circuits. For more information, see *Apply a Typical Circuit to an Item*, page 143.

Apply a Typical Schematic to an Item

Applying a typical schematic to a project item allows you to add a definition of a schematic to the selected project item. You can then generate a schematic without the need to define the schematic drawing properties, such as schematic blocks, templates, and so forth. For more information, see *Apply a Typical Schematic to an Item*, page 146.

Apply a Typical Control Station to an Item

Applying a typical control station to a project item allows you to create a predefined control station and associate it with a selected project item. The software allows you to apply a typical control station to all loads, all converting equipment items, disconnect equipment, generators, and battery banks. For more information, see *Apply a Typical Control Station to an Item*, page 144.

Apply a Set of Signals to an Item

Applying a typical I/O set to a project control station or a circuit creates predefined I/O signals and associates them with a selected item. You can then publish the signal data to SmartPlant Instrumentation. For more information, see *Apply a Set of Signals to an Item*, page 148.

Apply Lookup Table Data to Equipment

This procedure shows how to apply a lookup table to a project item. For more information, see *Apply Lookup Table Data to Equipment*, page 143.

Apply a Profile to an Item

A profile is an assembly of various typical items that you can use as a template for creating project items. You can apply a profile to any type of load and converting equipment. This procedure shows how to apply a profile to a project item. For more information, see *Apply a Profile to an Item*, page 147.

Related Topics
- *Apply Lookup Table Data to Equipment*, page 143

Apply Options to a Set of Tags

1. Do one of the following:

   - In the **Electrical Index**, select a set of tags. For more information, see *Select a Set of Tags*, page 41.

   - Select multiple tags in the Tabular Editor.

2. Click **Tools > Apply Options**.

3. On the **Apply Options** dialog box, select the values that you require on the appropriate tabs.

4. After you have selected the values on a tab, click **Apply**.

Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Apply a Typical Circuit to an Item*, page 143
   - *Apply a Typical Control Station to an Item*, page 144
   - *Apply Lookup Table Data to Equipment*, page 143
   - *Apply Typical Circuit Data to a Project Circuit*, page 173
   - *Create Load Items in Batch Mode*, page 131

Apply Lookup Table Data to Equipment

Caution

   - When you apply data using a lookup table, the software can overwrite existing data. Since this process is not reversible, you should proceed with great care.

   - It is good practice not to include calculated properties in lookup tables. For example, if the **Supply AC/DC Flag** property for a motor is set to **DC**, and you try to select a value for the **Frequency** property, you cannot apply this value and the software rejects the operation. You must first set **Supply AC/DC Flag** to **AC**, and only then you can apply the lookup table value for the **Frequency** property to the motor.

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder for the desired equipment type and do one of the following:

   - Expand the folder and select an item from the list.

   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

   Tip

   - You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Lookup Table** tab.

5. Select **Enable lookup table**.

   💡 **Tip**

   - The default lookup table for the equipment type appears in the **Lookup table** box. You can select a different profile by performing the following steps.

6. Beside the **Lookup table** box, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available lookup tables.

8. From the list of lookup tables, highlight the lookup table that you want to apply.

   💡 **Tip**

   - For each item property, the software overwrites existing values if the value of the **Overwrite Option** property in the lookup table is set to **True**. It is recommended that you view the lookup table before applying the data to verify which properties the software overwrites.

9. Click **OK**.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Applying Options Common Tasks*, page 141
- *Create Lookup Tables*, page 62
- *Creating Reference Data Common Tasks*, page 125
- *Generate Error Log Files*, page 120

Apply a Typical Circuit to an Item

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder for the equipment type that you require and do one of the following:

   - Expand the folder and select an item from the list.

   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

   💡 **Tips**

   - You can apply a typical circuit to all loads, all converting equipment items, and circuits.

   - You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Typical Circuit** tab.

5. Select **Enable typical circuit**.

6. Beside the **Typical circuit** box, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available typical circuits.

8. From the list of typical circuits, highlight the typical circuit that you want to assign to the equipment.

9. Click **OK**.

10. Under **Apply to**, select one of the following options according to the rule the software uses to apply the typical circuit:

    - **All items** - Applies the typical circuit to all the selected items.

    - **Items with no typical circuit** - Applies the typical circuit only to items without an existing typical circuit. Use this option to add a typical circuit only to new items among the items that you selected.

    - **Items with a different typical circuit from the selection** - Changes the typical circuit of the selected items to the current typical circuit. Use this option to add a typical circuit to items with a modified typical circuit definition.

    - **Items with the same typical circuit as the selection** - Updates an existing typical circuit for the selected items.

11. Click **Apply**.

Notes

- If you apply a typical circuit to an item that is already associated with a circuit, the software adjusts the components of the circuit to match the typical circuit specified by the item.

- If you apply a typical circuit that includes associated cables to a circuit, the software creates cables in the project and associates their 'To' sides with the circuit.

- If you modify the components of a typical circuit that already has a reference from an item, the software does not update the change in the reference until you next apply a typical circuit or a profile to the item.

- If the item that you are applying the typical circuit to is connected to a feeder circuit, the system replaces the circuit so that it matches the applied typical circuit as specified in the profile.

- If the item is connected to feeder equipment other than a circuit (such as a variable frequency drive, a transformer, or a battery bank), the software does not apply the typical circuit.

- If more than one item is connected to the same circuit, the software changes the last applied typical circuit for all the items.

- You can clear the typical circuit of an item that is not connected to any circuit by selecting **Clear the last applied typical circuit**.

- You can apply a typical circuit on a circuit, provided that the circuit has no equipment connected to it.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Create a Typical Circuit*, page 58
- *Creating Reference Data Common Tasks*, page 125
- *Generate Error Log Files*, page 120

Apply a Typical Control Station to an Item

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder for the desired equipment type and do one of the following:

   - Expand the folder and select an item from the list.

   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

   💡 **Tip**

   - You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Control Station** tab.

5. Select **Enable control station**.

6. Click **Add**.

7. On the **Find** dialog box, click **Find Now** to display the available control stations.

8. From the list of control stations, while holding **Ctrl**, highlight each control station that you want to associate with the item.

9. Click **OK**.

10. Under **Apply to**, click **All items**.

    💡 **Tip**

    - If required, you can click **Items with no control station** to add control stations only to items that do not have any existing control stations.

11. Under **Action**, click **Add** or **Replace** to determine how the software deals with control stations that have the same name as existing control stations. This option applies only if you click **All items**.

12. Click **Apply**.

📝 **Notes**

- You can apply a typical control station to a load, converting equipment item, disconnect electrical equipment item, a generator, and a battery bank.

- You can view the control stations associated with an item of electrical equipment by selecting **Common Properties** on the shortcut menu for the item, and on the dialog box that opens, selecting the **Controls & Signals** tab.

- If you select a reference control station that has one or more associated control cables, the software creates these cables in the control station that you add to the project item.

- If you apply control stations to one or more items that are connected to circuits, the software associates the control station cables (instrumentation and control) with the circuits automatically when you select the appropriate Options Manager setting. If you do not use the setting, you need to make the association manually. For details, see *Associate Control Station Cables with Equipment Circuits*, page 227.

- If you modify the components of a reference control station that already has a reference from an item, the software does not update the change in the reference until you next apply a profile to the item.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Associate a Control Station with an Item*, page 226
- *Create a Control Station*, page 176
- *Creating Reference Data Common Tasks*, page 125
- *Generate Error Log Files*, page 120

Apply a Typical Schematic to an Item

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** by clicking the + icons or by double-clicking the folders.

2. Select the **Motors**, **Static Electrical Equipment**, or **Other Electrical Equipment** folder and do one of the following:

   - Expand the folder and select an item from the list.

   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Typical Schematic** tab.

5. Select **Enable typical schematic**.

6. Beside the **Typical schematic** box, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available typical schematics.

8. From the list of typical schematics, highlight the typical schematic that you want to assign to the item.

9. Click **OK**.

10. Under **Apply to**, select one of the following options according to the rule the software uses to apply the typical schematic:

   - **All items** - Applies the typical schematic to all the selected items.

   - **Items with no typical schematic** - Applies the typical circuit only to items without an existing typical schematic. Use this option to add a typical circuit only to new items among the items that you selected.

   - **Items with a different typical schematic from the selection** - Changes the typical schematic of the selected items to the current typical circuit. Use this option to add a typical schematic to items with a modified typical circuit definition.

11. Click **Apply**.

📝 **Notes**

   - If you modify the components of a typical schematic that already has a reference from an item, the software does not update the change in the reference until you next apply a typical schematic or a profile to the item.

   - You can assign a new typical schematic or change an assigned typical schematic for an item at any time.

Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Create a Typical Circuit*, page 58
   - *Creating Reference Data Common Tasks*, page 125
   - *Generate Error Log Files*, page 120

Apply a Profile to an Item

🚩 **Caution**

   - When you apply a profile to an item, the software can overwrite existing data. Since this process is not reversible, you should proceed with great care.

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder for the desired equipment type and do one of the following:

   - Expand the folder and select an item from the list.

- Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

  💡 **Tip**

  - You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Equipment Profile** tab.

5. Select **Enable equipment profile**.

   💡 **Tip**

   - The default profile appears in the **Create item based on profile** box. You can select a different profile by performing the following steps.

6. Beside **Create item based on profile**, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available profiles.

8. From the list of profiles, highlight the profile that you want to apply.

9. Click **OK**.

10. Under **Apply to**, select one of the following options according to the rule the software uses to apply the profile:

    - **All items** - Applies the profile to all the selected items.

    - **Items with no profile** - Applies the profile only to items without an existing profile. Use this option to add a profile only to new items among the items that you selected.

    - **Items with a different profile from the selection** - Changes the profile of the selected items to the current profile. Use this option to add a profile to items with a modified profile definition.

    - **Items with the same profile as the selection** - Updates an existing profile for the selected items.

11. Select the **Populate empty item properties** check box to overwrite null data values for target item properties with values that exist in the profile. Clear the check box to leave the properties empty. (In all cases, the software does not overwrite existing values.)

12. Click **Apply**.

📝 **Notes**

- Applying a profile is a useful way of creating new items and populating large amounts of data in your project. When you apply the profile, the software automatically applies any definitions for typical circuits, control stations, typical I/O sets, and lookup tables that you define on the other tabs of the **Apply Options** dialog box.

- After you apply a profile for which the software creates new items and relations successfully, the operation may still partially fail, for example if certain values of updated data conflict with a rule. In such cases, the software displays an error message and records the problem in the log file.

- When you apply a profile, the software replaces all formerly associated items with the items specified in the current profile including any values that you have modified. The software assigns new item tags according to the naming convention.

- If you apply a profile that contains a reference item to one or more project items, the software populates the properties of those project items with values copied from the reference item.

- If you modify the components of a typical circuit that already has a reference from an item, the software does not update the change in the reference until you next apply a typical circuit or a profile to the item.

- When an item has associated cables, and you connect the item to a circuit by applying a profile with a typical circuit, if there is more than one cable, the software associates the free ends of all the cables to the circuit.

- If you apply a profile for a reference motor with a heater to a project motor without a heater, the software adds a heater to the project motor.

- If you apply a profile for a transformer, the software applies the profile only if the reference and project transformers have the same number of windings and where the **Number of Secondaries** property for the project transformer has a value of **1** or **2**. If no value appears for this property, you must type one manually in accordance with the number of secondary windings in the transformer.

- If the item that you are applying a profile to is connected to a feeder circuit, the system replaces the circuit so that it matches the applied typical circuit as specified in the profile.

- If the item is connected to feeder equipment other than a circuit (such as a variable frequency drive, a transformer, or a battery bank), the software does not apply the profile.

Related Topics

Apply a Set of Signals to an Item

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Navigate to the folder for the desired equipment type and do one of the following:

   - Expand the folder and select an item from the list.

   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

   💡 **Tip**

   - You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Typical I/O Sets** tab.

5. Select **Enable typical I/O sets**.

6. Beside the **Typical I/O sets** box, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available set of signals.

8. From the list of signal sets, highlight the set of signals that you want to assign to the item.

9. Click **OK**.

10. Under **Apply to**, select one of the following options according to the rule the software uses to apply the set of signals:

    - **All items** - Applies the set of signals to all the selected items.

    - **Items with no typical I/O sets** - Applies the set of signals only to items without an existing set of signals. Use this option to add a set of signals only to new items among the items that you selected.

    - **Items with a different typical I/O set from the selection** - Changes the set of signals of the selected items to the current set of signals. Use this option to add a set of signals to items with a modified typical I/O set definition.

11. Click **Apply**.

📝 **Notes**

- If you modify the components of a set of signals that already has a reference from an item, the software does not update the change in the reference until you next apply a set of signals or a profile to the item.

- You can assign a new set of signals or change an assigned set of signals for an item at any time.

Managing Power Distribution Boards Common Tasks

The following tasks are used frequently when you manage power distribution boards (PDBs).

📝 **Note**

- In the **Reference Data Explorer**, you can create PDBs with buses only.

Create a Power Distribution Board (PDB)

This procedure explains how to create a new power distribution board (PDB) in the **Electrical Index**. For more information, see *Create a Power Distribution Board (PDB)*, page 152.

Create a Bus

This procedure explains how to add a new bus to a power distribution board. For more information, see *Create a Bus*, page 154.

Create a Stand-Alone Bus

SmartPlant Electrical allows you to create stand-alone buses that are not associated with power distribution boards. You create these buses in the **Substations** folder of the **Electrical Index**. Stand-alone buses can be parent items of circuits and circuit internals, but not cells. For more information, see *Create a Stand-Alone Bus*, page 155.

Create a Cell

This procedure shows how to create a new cell. For more information, see *Create a Cell*, page 155.

Working with Circuits Common Tasks

This topic describes various procedures dealing with creation of circuits and their internal components. For more information, see *Working with Circuits Common Tasks*, page 170.

Delete a PDB or a PDB Internal Item

This procedure shows how to delete a power distribution board or any PDB internal item. The topic also states the deletion preconditions and lists the other items that the software deletes with the selected item. For more information, see *Delete a Power Distribution Board or a PDB Internal Item*, page 156.

Design a PDB Layout

This procedure shows you how to design a power distribution board layout for PDBs that contain cells. When designing a layout for a power distribution board, you define its internal structure by creating *compartments* and assigning the cells to the compartments. After designing a PDB layout, you can generate and print out the PDB layout report in the format of an Excel spreadsheet. For more information, see *Design a PDB Layout*, page 158.

Resize a Cell

This procedure explains how to resize a cell when designing a PDB layout. The software allows you to define compartments that can accommodate cells that are bigger or smaller than the default cell size. You can create as many compartments as you need. The maximum number of possible compartments depends on the values that you enter for the **Compartments in Row** and **Compartments in Section** properties in the **Properties** window. For more information, see *Resize a Cell on the PDB Layout Design Sheet*, page 162.

Change Cell Assignment

This procedure shows you how to assign a cell to another *compartment* when designing a PDB layout. The software allows you to change cell assignment as long as the target compartment is not occupied by another cell and that the target compartment fits the size of the cell. For more information, see *Change Cell Assignment to Another Compartment*, page 164.

Cancel the Assignment of a Cell to a Compartment

This procedure shows you how to cancel a cell assignment when designing a PDB layout. For more information, see *Cancel the Assignment of a Cell to a Compartment*, page 162.

Generate a PDB Layout Design Report

This procedure explains how to generate and print out a PDB Layout Design Report. For more information, see *Generate a PDB Layout Design Report*, page 164.

Create a Power Distribution Board (PDB)

1. In the **Electrical Index** or the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment**.

2. Right-click the **Power Distribution Board** folder and click **New**.

3. Accept the default item tag or type your own item tag for the PDB.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string of up to 80 characters in length.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the PDB. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the PDB properties as desired in the **Properties** window.

   💡 **Tip**

   - You should enter the values needed for PDB design in the **Properties** window right after the creation of a new power distribution board. These properties are essential to PDB design and you should define them at this initial stage. Changing the property settings later is likely to become problematic. For example, if you do not define the number of horizontal rows and the number of vertical sections of the PDB cabinet, the software applies the default values to define the physical dimensions of the cabinet. Changing the physical dimension properties during the PDB layout design results in the loss of cells and circuitry. For more details, see *Power Distribution Board Properties*, page 159.

   📝 **Note**

   - You can also create a power distribution board by dragging a typical PDB from the **Reference Data Explorer** to the **Electrical Index**.

   - You can customize the appearance of how the PDB appears on your SLD drawings. The options available include line style, width, and color, position of SmartText and so forth. For more details, see Creating Symbols: An Overview in the *Catalog Manager User's Guide*, under *Catalog Manager*, *Working with Symbols*, *Creating Symbols*.

   Related Topics
   - *Design a PDB Layout*, page 158
   - *Managing Power Distribution Boards Common Tasks*, page 151
   - *Power Distribution Board Properties*, page 159

Create a Bus

1. In the **Electrical Index** or the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards**.

2. Click a PDB to expand its folder hierarchy.

3. Right-click the **Buses** folder and click **New**.

4. Accept the default item tag or type your own item tag for the bus.

   💡 **Tips**

   - In the **Reference Data Explorer**, you can type a string up to 80 characters long.

   - In the **Electrical Index**, the name must be in accordance with the *naming convention* for the bus. For more details of naming convention settings, see *Using Naming Conventions*, page 127.

5. Edit the bus properties as required in the **Properties** window.

   📝 **Note**

   - You can also create a dual bus. Create buses and then connect them using a bus tie. For information about bus tie creation, see *Associate Bus Riser Circuits with Coupler Circuits*, page 226.

   Related Topics
   - *Create a Cell*, page 155
   - *Create a Power Distribution Board (PDB)*, page 152
   - *Managing Power Distribution Boards Common Tasks*, page 151

Create a Stand-Alone Bus

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Substations**.

2. Right-click the **Substations** folder and click **New Bus**.

   📝 **Note**

   - The software creates a new stand-alone bus and names it automatically. In the **Electrical Engineer**, stand-alone buses appear at the top level.

   Related Topics
   - *Create a Circuit*, page 171
   - *Create Circuit Components*, page 173
   - *Managing Power Distribution Boards Common Tasks*, page 151

Create a Cell

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards** > **Buses**.

2. Right-click a bus and click **New Cell**.

   💡 **Tip**

   - As an alternative method, you can create a cell by right-clicking the **Cells** folder and selecting **New Cell**. In this case, you create a free cell that is not connected to any power source and that does not appear in the **Electrical Engineer**. You cannot create any circuits in a free cell.

3. Accept the default item tag that is based on the *naming convention* for the cell, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the other cell properties in the **Properties** window as required.

📝 **Notes**

- The software specifies the cell function (feeder, incomer, *coupler*, or *bus riser*) in the **Properties** window after you add a circuit to a cell. The software validates that the cell function is compatible with the circuits that it contains. If you add a circuit with a function type that does not match the function of the current cell, the software changes the cell function in the **Properties** window automatically to make these functions compatible.

- To include the current cell in the PDB schedule report, in the **Properties** window, select **In Use** when you define the **Cell Usage** property.

- You cannot specify the **Cell Height**, **Cell Width**, and **Cell Index** properties in the **Properties** window at this stage as these properties are calculated and displayed by the software after you assign a cell to a compartment when designing a PDB layout. For more information, see *Design a PDB Layout*, page 158.

Related Topics
- *Create a Bus*, page 154
- *Create a Power Distribution Board (PDB)*, page 152
- *Managing Power Distribution Boards Common Tasks*, page 151

Delete a Power Distribution Board or a PDB Internal Item

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards**.

2. Select a PDB or one of its internal items.

3. Click **Edit > Delete**.

   💡 **Tip**

   - You can also delete a PDB or any PDB internal item by right-clicking an item and selecting **Delete** on the shortcut menu.

   📝 **Notes**

   - You can delete only items that do not have associations to other items. For example, you cannot delete a power distribution board associated with loads unless you first dissociate the loads.

   - If you select more than one item for deletion, the software records any deletion failures in the **Batch Deletion** log file. For the procedure for creating and viewing the log file, see *Generate Error Log Files*, page 120.

Deletion Rules for PDBs and PDB Internal Items

The following rules apply when you delete a PDB or a PDB internal item in the **Electrical Index**:

PDB Item	Precondition	Result
Power distribution board	You have dissociated all the circuits in the PDB from external items, such as cables, loads, and so forth.	The software also deletes all the internal buses, cells, circuits, and circuit components.
Bus	You have dissociated all the bus circuits from external items, such as cables, loads, and so forth.	The software also deletes all the cells, circuits, and circuit components.
Cell	You have dissociated all the circuits in the cell from external items, such as cables, loads, and so forth.	The software also deletes all the circuits in the cell and the circuit components.
Circuit	You have dissociated all the external items, such as cables, loads from the circuit.	The software also deletes all the circuit components.
Circuit internal		The software deletes the circuit internal and clears any existing association with a typical circuit.

Related Topics
- *Create a Bus*, page 154
- *Create a Cell*, page 155
- *Create a Circuit*, page 171
- *Create a Power Distribution Board (PDB)*, page 152
- *Create Circuit Components*, page 173
- *Managing Power Distribution Boards Common Tasks*, page 151

Design a PDB Layout

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards**.

    💡 **Tips**

    -   The physical structure of a power distribution board depends on the values you enter in the **Properties** window for the selected PDB. These properties define the size of the PDB (the number of vertical sections and horizontal rows) and the PDB internal structure (the number of *compartments*). The default physical arrangement of PDB cabinets is five horizontal rows and five vertical sections. If you wish to have a different physical arrangement, enter the desired values in the **Properties** window before designing the PDB layout. For more information, see *Power Distribution Board Properties*, page 159. Changing an existing physical arrangement after you assign the cells on the PDB layout design sheet results in the loss of cell assignment on the PDB layout design sheet.

    -   Note that you can design PDB layouts only for power distribution boards that you created with cells.

2. Right-click a PDB and click **Design PDB Layout**.

3. Click the **Buses** folder to expand its hierarchy.

4. Click the desired bus to display its cells.

5. Highlight a cell and drag it to the desired compartment on the displayed layout design sheet.

    💡 **Tip**

    -   Make sure that you create all the cells that you require and define their properties in the **Properties** window before designing the PDB layout. This way the compartments on the layout design sheet display the item tags and the PDB layout report shows the cell circuitry, cell connections, functions, and usage. Also, note that the software automatically displays **Cell Height**, **Cell Width**, and **Cell Index** value properties in the **Properties** window for the current cell after you assign a cell to a compartment. The software calculates these values in accordance with the number of compartments in rows and sections for the current compartment.

Now you can do the following:

- Resize a cell. (You can also resize a cell before you assign it to a cell box.) The maximum number of horizontal and vertical compartments that you can have in a cell depends on the values you entered in the **Compartments in Row** and **Compartments in Section** in the **Properties** window for the current PDB. For more information, see *Power Distribution Board Properties*, page 159.

- Change or cancel the assignment of a cell.

- Change the magnification level of the PDB design sheet by zooming in or out.

- Generate a report that shows the layout design sheet and all the contents of the cells in the current PDB.

Related Topics
- *Cancel the Assignment of a Cell to a Compartment*, page 162
- *Change Cell Assignment to Another Compartment*, page 164
- *Change the Zoom Level in a PDB Layout Design Sheet*, page 164
- *Generate a PDB Layout Design Report*, page 164
- *Managing Power Distribution Boards Common Tasks*, page 151
- *Open a PDB Layout Design for a Selected PDB*, page 161
- *Resize a Cell on the PDB Layout Design Sheet*, page 162

Power Distribution Board Properties

The following table describes the essential power distribution board properties. These properties define the physical structure of a PDB cabinet. You must define these properties before you start designing a layout for a power distribution board. Changing the physical structure of a power distribution board for which a PDB layout design already exists restructures all the *compartments* and cancels cell assignment to the compartments.

Property	Description
Number of Rows	The number of horizontal rows that a PDB will contain. The default value is 5. The maximum is 15 rows.
Number of Sections	The number of vertical sections that a PDB will contain. The default value is 5. The maximum is 30 sections.
Compartments in Row	The number of possible compartments that a particular horizontal row can contain. The default value is 1 (this means that this cell can have only one horizontal compartment and it cannot be divided horizontally). The maximum is 4 compartments in one row.
Compartments in Section	The number of possible compartments that a particular vertical section can contain. The default value is 1 (this means that this cell can have only one vertical compartment and it cannot be divided vertically). The maximum is 4 compartments in one section.
Section Constant String	When you select **Section Naming Type** as **Constant String + Ascending Number**, you specify the constant string value by this property.
Section Naming Type	The vertical section naming convention, for example: Alphabetic (A, B, C...), Numeric (1, 2, 3...), or Constant String + Ascending Number.
Section Starting Number	The number from which you start numbering the vertical sections. Applicable only for sections that use a numeric naming convention.
Origin Point	The compartment layout orientation in a power distribution board. For example, top left -> bottom right. This means that the compartments are laid out from top to bottom and from left to right.
Row Constant String	When you select **Row Naming Type** as **Constant String + Ascending Number**, you specify the constant string value by this property.
Row Naming Type	The horizontal row naming convention, for example: Alphabetic (A, B, C...), Numeric (1, 2, 3...), or Constant String + Ascending Number.
Row Starting Number	The number from which you start numbering the horizontal rows. Applicable only for rows that use a numeric naming convention.

Open a PDB Layout Design for a Selected PDB

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards**.

2. Right-click the required PDB and click **Open PDB Layout**.

Related Topics
- *Cancel the Assignment of a Cell to a Compartment*, page 162
- *Change Cell Assignment to Another Compartment*, page 164
- *Change the Zoom Level in a PDB Layout Design Sheet*, page 164
- *Design a PDB Layout*, page 158
- *Generate a PDB Layout Design Report*, page 164
- *Managing Power Distribution Boards Common Tasks*, page 151
- *Resize a Cell on the PDB Layout Design Sheet*, page 162

Resize a Cell on the PDB Layout Design Sheet

1. In the PDB layout design sheet, right-click a *compartment*.

   💡 **Tip**

   - When increasing the size of a compartment, make sure that you divide the adjacent compartment that will accommodate the new increased cell before you start increasing the size of the current compartment.

2. On the shortcut menu, click **Change Cell Size**.

3. On the **Cell Size** dialog box, do the following:

   - Enter the new width. For example, 0.5, 1, 1.5, etc.

   - Enter the new height. For example, 0.5, 1, 1.5, etc.

   💡 **Tips**

   - You can also resize a compartment by dragging it to another unassigned compartment.

   - You can resize a compartment that is already assigned to a cell.

📝 Notes

- One unit of measure represents the width or height of a standard cell. You can make the size of a cell as big as needed as long as the cell is within the limits of the defined PDB cabinet. Also, you can define a cell to have a fraction of a cell size. For example, enter 0.5 for half the cell width or height.

- When decreasing the size of a compartment, the maximum number of compartments that a vertical section or a horizontal row can have depends on the value that you enter for the **Compartments in Row** and **Compartments in Section** properties in the **Properties** window for the power distribution board to which the cell belongs.

- When increasing the size of a compartment, the software enlarges the compartment only to its right or down.

- When increasing the size of a compartment, the software checks if the adjacent cells that are under and to the right of the current cell can accommodate the new cell size. Note that you have to divide the adjacent compartment to prepare it for the new increased cell.

- You cannot exceed the boundaries of a vertical section. You can only increase the number of compartments within the boundaries of a vertical section. You can increase the height of a compartment. In this case, the compartment will occupy the full size of the compartment located directly below it.

- The software resizes the compartments with the upper left corner of the compartment as its point of origin.

- You cannot resize a compartment if its new size infringes on an adjacent compartment that is occupied by another cell.

Related Topics

Cancel the Assignment of a Cell to a Compartment

1. On the PDB layout design sheet, right-click the *compartment* for which you want to cancel the cell assignment.

2. Click **Cancel Assignment**.

Change Cell Assignment to Another Compartment

- In the PDB layout design sheet, drag the desired cell to another *compartment*.

📝 **Notes**

- If you reassign a cell to a compartment that has a cell size different from the cell size of the current cell, the software resizes the target compartment to accommodate the cell you are reassigning.

- You cannot reassign a cell if the software cannot resize the target compartment.

- You cannot reassign a cell if the target compartment is occupied by another cell.

Related Topics
- *Cancel the Assignment of a Cell to a Compartment*, page 162
- *Change the Zoom Level in a PDB Layout Design Sheet*, page 164
- *Design a PDB Layout*, page 158
- *Generate a PDB Layout Design Report*, page 164
- *Managing Power Distribution Boards Common Tasks*, page 151
- *Resize a Cell on the PDB Layout Design Sheet*, page 162

Change the Zoom Level in a PDB Layout Design Sheet

1. Right-click the PDB layout design sheet.

2. On the shortcut menu, click **Zoom**.

3. On the **Zoom Options** dialog box, do one of the following:

- Display the full cabinet size.

- Select a custom zoom option.

- Use the default cabinet size.

Generate a PDB Layout Design Report

1. Right-click the PDB layout design sheet.

2. On the shortcut menu, click **Generate Report**.

📝 **Notes**

- Make sure that you create all the desired cells and define their properties in the **Properties** window before designing the PDB layout and generating its report. This way the PDB layout design report shows the cell circuitry, cell connections, functions, and usage.

- The report is generated as an Excel workbook with the PDB layout sheet shown on the first sheet. The cell and circuit details appear on subsequent sheets.

Related Topics

- *Cancel the Assignment of a Cell to a Compartment*, page 162
- *Change Cell Assignment to Another Compartment*, page 164
- *Change the Zoom Level in a PDB Layout Design Sheet*, page 164
- *Design a PDB Layout*, page 158
- *Managing Power Distribution Boards Common Tasks*, page 151
- *Resize a Cell on the PDB Layout Design Sheet*, page 162

Using Dual Source Power Supplies

SmartPlant Electrical allows you to specify two power sources for equipment: a primary source and an alternative source. Examples of dual source power supplies include:

- A motor fed by supplies on two separate buses.

- A variable-frequency drive fed by the two secondaries of a 3-winding transformer.

- An uninterruptible power supply fed by a main input with battery back-up.

To set up a dual power supply, you first create each of the feeder branches and associate the main supply with the item. Next, you specify the second feeder branch as the alternative power supply. The software can display dual power supplies in single line diagrams.

📝 Notes

- Loads that are connected to circuits for which the circuit mode value is **Disconnected** appear in load summaries, but the software does not take those loads into account in the total load calculation. If you do not specify circuit mode value **Disconnected**, the software may count the load twice in load summary reports.

- For batch load association calculations, in normal mode, the software only takes into account the main supply. In "drill down" mode, the software takes into account both sources, provided you did not specify the circuit mode value as **Disconnected**.

Associate a Dual Power Supply with an Item

1. In the **Electrical Index**, create the load to which you want to associate the power supplies.

2. Create two power cables to associate with the item.

3. Create two feeder branches: one for each supply, for example a generator and an off-site power supply, each attached to a bus.

4. In the **Electrical Engineer**, associate the load with the main supply. For details, see *Associate Electrical Equipment with Feeders*, page 216.

5. Select the load and right-click.

6. On the shortcut menu, click **Select Alternative Power Source**.

7. On the dialog box that opens, navigate to the item that you want to designate as the alternative supply.

    ### 💡 Tip

    -   If the item is not visible in the window, select a branch and click **Expand**. Alternatively, click **Find** to locate the item using the **Find** dialog box.

8. Click **OK** to select the item as the alternative power supply.

    ### 💡 Tip

    -   The **OK** button is only enabled when you select an item that is valid for use as an alternative power supply.

9. To view the load power supplies, select the load, and on the shortcut menu, click **Common Properties**.

10. On the **Common Properties** dialog box for the load, click the **Feeder Data** tab to view details of the main power supply and click the **Alternative Feeder Data** tab to view the name of the feeder equipment used as the secondary power supply.

📝 Notes

-   In the **Electrical Engineer**, the software places a visual indication '(#)' beside the names of the load and the feeder cable where they appear in the alternative power supply branch.

-   You can change the alternative power source by right-clicking the feeder cable in the alternative power supply branch and clicking **Select Alternative Power Source**.

-   If the load is only connected to an alternative power source, any drag-and-drop operations that you perform in the **Electrical Engineer** affect the primary power source only.

Related Topics
-   *Dissociate an Alternative Power Supply from an Item*, page 168
-   *Using Dual Source Power Supplies*, page 165

Change an Alternative Power Supply to an Item

1. In the **Electrical Engineer**, select the load for which you want to change the alternative supply.

   💡 **Tip**

   - You can only change the alternative supply if you select the item at the lowest level of the branch.

2. Right-click the load.

3. On the shortcut menu, click **Change Alternative Power Source**.

4. On the dialog box that opens, navigate to the item that you want to designate as the alternative supply.

   💡 **Tip**

   - If the item is not visible in the window, select a branch and click **Expand**. Alternatively, click **Find** to locate the item using the **Find** dialog box.

5. Click **OK** to select the item as the alternative power supply.

   Related Topics
   - *Associate a Dual Power Supply with an Item*, page 166
   - *Using Dual Source Power Supplies*, page 165

Dissociate an Alternative Power Supply from an Item

1. In the **Electrical Engineer**, select the load for which you want to dissociate the alternative supply.

   💡 **Tip**

   - You can only dissociate the alternative supply if you select the item at the lowest level of the branch.

2. Click **Actions** > **Dissociate**.

   Related Topics
   - *Associate a Dual Power Supply with an Item*, page 166
   - *Change an Alternative Power Supply to an Item*, page 166
   - *Using Dual Source Power Supplies*, page 165

Using Transfer Switches

Transfer switches are two-way manual or automatic switches that are used to switch or transfer an essential load from the main power source to a backup power source.

In the **Electrical Engineer**, the software displays a transfer switch as a normal bus with two incomers. You drag each incomer separately to its parent power source. The software shows the two paths separately, with the items downstream of the transfer switch repeated on each path. The software can display transfer switches and their electrical paths in single line diagrams.

📝 Notes

- You cannot select transfer switches as designated PDBs on the **Feeder** tabs of the **Common Properties** dialog boxes for loads, converting equipment, and so forth.

- The software includes the name of a transfer switch on the **Feeder** tab for a connected load as part of the feeder path.

- After you create a transfer switch, you can add cells and circuit internals to it.

Related Topics
- *Create a Transfer Switch*, page 169
- *Generate an SLD for a Transfer Switch*, page 169

Create a Transfer Switch

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards** > **Transfer Switches** folder.

2. Right-click the **Transfer Switches** folder, and on the shortcut menu, click **New Transfer Switch**.

   ### 💡 Tip

   - The software creates a PDB, a bus, two incomers, and a feeder. The **Equipment Type** property for the PDB is set as **Transfer Switch**.

3. Select one of the incomers.

4. Right-click the incomer, and on the shortcut menu, click **Common Properties**.

5. On the **Circuit Common Properties** dialog box, select a value for the **Circuit Mode** property: **Connected** or **Disconnected**.

💡 **Tip**

- When you select a circuit mode value **Connected** for a transfer switch incomer, the value for the second incomer circuit mode automatically changes to **Disconnected**. You can set both incomers to be disconnected, or one only.

6. Specify values for other incomer properties as you require.

7. Close the dialog box and in the **Electrical Engineer**, connect items to the feeder and incomer branches as you require.

Related Topics
- *Associating Electrical Equipment Common Tasks*, page 215
- *Generate an SLD for a Transfer Switch*, page 169
- *Using Transfer Switches*, page 168

Generate an SLD for a Transfer Switch

1. In the **Electrical Engineer**, while holding down the **Ctrl** key, select the top level items in both of the branches associated with the transfer switch.

2. Right-click one of the selected items, and on the shortcut menu, click **New SLD**.

Related Topics
- *Create a Transfer Switch*, page 169
- *Using Transfer Switches*, page 168

Working with Circuits Common Tasks

The following tasks are used frequently when you add feeder circuits to buses on power distribution boards (PDBs). Usually, you define your circuits and then assign them to loads.

Create a Circuit

This procedure shows how to create a new circuit in the **Electrical Index**. For more information, see *Create a Circuit*, page 171.

Copy Circuit Components

This procedure shows how to copy circuit components, such as protection devices, disconnect devices, electrical distribution equipment, and so forth, from the **Reference Data Explorer**. For more information, see *Copy Circuit Components*, page 172.

Apply Typical Circuit Data to a Project Circuit

This procedure shows how to apply circuit components using a typical circuit from the **Reference Data Explorer** to an existing project circuit   This method is recommended for bulk replacement of circuits with internal components. For more information, see *Apply Typical Circuit Data to a Project Circuit*, page 173.

Create Circuit Components

This procedure shows how to add individual circuit components, such as protection devices, disconnect devices, electrical distribution equipment, and so forth, to a circuit. For more information, see *Create Circuit Components*, page 173.

Change the Location of a Circuit on a Bus

This procedure shows how to move a circuit between cells on a bus or between a cell on a bus and the bus itself. For more information, see *Change the Location of a Circuit on a Bus*, page 175.

Related Topics
- *Associate Electrical Equipment with Feeders*, page 216
- *Associate Loads with a PDB in Batch Mode*, page 219

Create a Circuit

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards** > **Buses**.

   💡 **Tip**

   - You can also create a circuit by dragging a typical circuit from the **Reference Data Explorer** to the **Electrical Index**.

2. Right-click a bus and on the shortcut menu, select one of the following:

   - **New Feeder Circuit** 🔲
   - **New Incomer Circuit** 🔲
   - **New *Coupler Circuit*** 🔲
   - **New *Bus Riser Circuit*** 🔲

3. Accept the default item tag that is based on the *naming convention* for the circuit, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. Edit the circuit properties as required on the **Circuit Common Properties** dialog box.

   ✍ **Notes**

   - You can associate the internal components of a circuit with relevant lookup tables that contain appropriate data. On the **Circuit Common Properties** dialog box, click the **Associate Items** tab. Then, in the **Circuit internals** data window, under **Lookup Table**, click and select the lookup table that you require. You will then be able to apply this associated lookup table to a project circuit when you run the **Apply Lookup Tables** feature on the **Apply Options** dialog box to propagate the lookup table data to the circuit internal components.

   - You cannot change the circuit function in the **Properties** window. If you need to change the function of an existing circuit, create a new one then delete the circuit for which the function is not appropriate.

   - You cannot add a circuit that has a function type that does not match the circuit function of the existing circuits belonging to the same bus or cell.

   Related Topics
   - *Associate Bus Riser Circuits with Coupler Circuits*, page 226
   - *Create a Power Distribution Board (PDB)*, page 152
   - *Managing Power Distribution Boards Common Tasks*, page 151

Copy Circuit Components

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards > Buses**.

2. Double-click a bus to expand its folder hierarchy.

3. Select the desired circuit.

4. In the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Disconnect Electrical Equipment**.

5. Select the appropriate sub-folder and drag the desired item to the circuit in the **Electrical Index**.

   📝 **Notes**

   - If the circuit was previously populated with typical circuit data, the software clears the reference to the typical circuit.

   - If the circuit is already associated with a load, the software clears the reference to the typical circuit.

   Related Topics
   - *Create a Circuit*, page 171
   - *Create Circuit Components*, page 173

Apply Typical Circuit Data to a Project Circuit

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards > Buses**.

2. Expand the **Buses** folder and in the lower pane, select one or more circuits to which you want to assign the typical circuit.

   💡 **Tip**

   - You can assign a typical circuit to a bus, in which case the software adds to the bus a new circuit containing components that are derived from the typical circuit.

3. Click **Tools > Apply Options**.

4. On the **Apply Options** dialog box, click the **Typical Circuit** tab.

5. Select **Enable typical circuit**.

6. Beside the **Typical circuit** box, click **Find**.

7. On the **Find** dialog box, click **Find Now** to display the available typical circuits that match the type of circuit you have selected to assign the data to, such as feeder or incomer.

8. From the list of typical circuits, highlight the typical circuit that you want to assign.

9. Click **OK**.

📝 **Notes**

- If you apply a typical circuit to more than one circuit in the **Electrical Index**, the software applies the typical circuit only to those circuits of the same type as the typical circuit.

- If you apply a typical circuit to a circuit that already has associated components, the software prompts you as to whether you want to replace the existing components with components belonging to the typical circuit.

- If you apply a typical circuit that includes associated reference cables, the software creates project cables from them and adds the project cables to the circuit with the **To** sides of the cables associated with the circuit.

- You cannot assign a typical circuit to a circuit that is already associated with a load.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Create a Typical Circuit*, page 58
- *Creating Reference Data Common Tasks*, page 125

Create Circuit Components

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards** > **Buses**.

2. Double-click a bus to expand its folder hierarchy.

3. Right-click the desired circuit and on the shortcut menu, click an option to create new disconnect, protection, converting or control equipment.

   💡 **Tip**

   - As an alternative method, you can create a circuit component by dragging from the **Reference Data Explorer** an item under **Disconnect Electrical Equipment**. For more information, see *Copy Circuit Components*, page 172.

4. Accept the default item tag that is based on the *naming convention* for the component, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

5. Edit the component properties as required in the **Properties** window.

✎ **Note**

- You cannot change the electrical equipment class and sub-class values in the **Properties** window. If you need to change these values, create a new component and then delete the component that is not needed.

Related Topics
- *Copy Components to a Typical Circuit*, page 59
- *Create a Circuit*, page 171
- *Managing Power Distribution Boards Common Tasks*, page 151

Add New Internal Item Dialog Box

Allows you to add an internal item to a PDB circuit from the **Circuit Common Properties** dialog box. Internal items can include various types of disconnect equipment, converting equipment, and measuring equipment.

Available Items - Allows you to select an item to add as an internal item for the circuit.

Change the Location of a Circuit on a Bus

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Power Distribution Boards** > **Buses**.

2. Expand the desired bus and locate the circuit you want to move.

3. Do one of the following:

   - Drag a circuit on a bus to a cell under the bus.

   - Drag a circuit in a cell to the bus on which the cell is located.

✎ **Notes**

- The circuit function type must match the function type of the bus or cell. For example: incomer or feeder.

- You can relocate a circuit within a specific bus only; you cannot move a circuit to a different bus.

Related Topics
- *Create a Bus*, page 154
- *Create a Cell*, page 155
- *Create a Circuit*, page 171
- *Managing Power Distribution Boards Common Tasks*, page 151

Creating Project Panels

You can create the following types of panels in the **Electrical Index**:

- **Control Stations** - Small cabinets containing low voltage start-stop circuitry used to control a load at a remote location. For more information, see *Create a Control Station*, page 176.

- **Local Panels** - Cabinets containing circuit, equipment, and wiring items, used for example, to provide switching or isolation close to a motor. For more information, see *Create a Local Panel*, page 177.

- **Junction Boxes** - Cabinets used to connect wires that run between various items of electrical equipment. For more information, see *Create a Junction Box*, page 178.

📝 **Note**

- In contrast to *Power Distribution Boards (PDBs)*, panels are much smaller in size and do not contain *busbars*.

Related Topics
- *Create a Panel Circuit*, page 179
- *Creating Reference Data Common Tasks*, page 125
- *Creating Typical Panels*, page 52

Create a Control Station

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Control Stations** folder, and then click **New**.

3. Accept the default item tag that is based on the *naming convention* for the control station, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. On the **Control Station Common Properties** dialog box, edit the control station properties as required. For more details, see *Edit Control Station Properties*, page 176.

📝 **Notes**

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you create a new control station.

- You access the **Control Station Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create a new control station.

Related Topics
- *Copy a Reference Item to the Electrical Index*, page 139
- *Creating Project Panels*, page 175
- *Working with SmartPlant Electrical Common Tasks*, page 34

Edit Control Station Properties

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels** > **Control Stations**.

2. Right-click the desired control station and on the shortcut menu, click **Common Properties**.

3. On the **Control Station Common Properties** dialog box, beside **Associated load**, click **Find** to open the **Find** dialog box.

4. From the **Equipment type** list, select the desired load type.

5. Click **Find Now**.

6. From the data window, highlight an item tag.

7. Click **OK** to associate the item tag with the current control station.

8. To add one or more control, grounding, or instrumentation cables to the control station, do the following:

   - Click **Add**.

   - On the **Find** dialog box, click **Find Now** to display the available control and grounding cables that have at least one unassociated side.

   - For each cable that you want to associate with the control station, select the cable and then click **Apply**.

9. To add an I/O signal to the control station that is going to be associated with a load, in the **Electrical Index**, right-click the control station, and on the shortcut menu, click **New Signal**.

Related Topics
- *Copy a Reference Item to the Electrical Index*, page 139
- *Creating Project Panels*, page 175
- *Working with SmartPlant Electrical Common Tasks*, page 34

Create a Local Panel

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Local Panels** folder and then click **New Local Panel**.

3. Accept the default item tag that is based on the *naming convention* for the panel, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. If required, right-click the local panel and click **New Circuit** to add a panel circuit.

   💡 **Tip**

   - Also, you can add a panel circuit by expanding the **Typical Circuits** folder in the **Reference Data Explorer** and dragging an existing typical panel circuit 📄 to the panel.

5. Right-click the desired circuit and on the shortcut menu, click an option to create new disconnect, protection, converting or control equipment.

6. Repeat the previous step as often as needed to create additional items for the circuit.

Related Topics
- *Create Circuit Components*, page 173
- *Creating Project Panels*, page 175

Create a Junction Box

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Junction Boxes** folder and then click **New Junction Box**.

3. Accept the default item tag that is based on the *naming convention* for the junction box, or type your own item tag according to the naming convention.

   💡 **Tip**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

4. If required, right-click the junction box and click **New Circuit** to add a panel circuit.

   💡 **Tip**

   - Also, you can add a panel circuit by expanding the **Typical Circuits** folder in the **Reference Data Explorer** and dragging an existing typical panel circuit 🔲 to the junction box.

5. Right-click the circuit and on the shortcut menu, click an option to create new disconnect, protection, converting or control equipment.

6. Repeat the previous step as often as needed to create additional items for the circuit.

Related Topics
- *Create Circuit Components*, page 173
- *Creating Project Panels*, page 175

Create a Panel Circuit

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels**.

2. Right-click the **Local Panels** folder or the **Junction Boxes** folder and select an item.

3. Right-click the local panel or the junction box where you want to add the circuit and click **New Circuit**.

   💡 **Tip**

   • Also, you can add a panel circuit by expanding the **Typical Circuits** folder in the **Reference Data Explorer** and dragging an existing typical panel circuit 📃 to the local panel or the junction box.

4. Accept the default item tag that is based on the *naming convention* for the circuit, or type your own item tag according to the naming convention.

   💡 **Tip**

   • For more details of naming convention settings, see *Using Naming Conventions*, page 127.

5. Right-click the desired circuit and on the shortcut menu, click an option to create new disconnect, protection, converting or control equipment.

6. Repeat the previous step as often as needed to create additional items for the circuit.

Related Topics
   • *Create a Typical Circuit*, page 58
   • *Create Circuit Components*, page 173
   • *Creating Reference Data Common Tasks*, page 125

Managing Cables: An Overview

You can create cables in your project for several purposes. SmartPlant Electrical specifies the following cable categories according to the purpose of the cable: power, control, grounding, and instrumentation. The software supports compliance with the strict regulations governing cable definitions by using standard reference cables as a basis for all the cables that you create in your project. Reference cables contain technical cable data such as voltage rating, ampacity, resistivity, cable material, and data relating to the construction of the cable. For this reason, you must initially create all your reference cables in the **Reference Data Explorer**.

The software allows you to organize your reference cables in families known as cable specifications. When you select a reference cable to use as a basis for your project cable, you must select the appropriate cable category and cable specification for the reference cable.

Also, when you size project cables, the software looks for the appropriate reference cable based on a particular cable specification and conductor arrangement.

Related Topics
- *Managing Cables Common Tasks*, page 181

Managing Cables Common Tasks

The following tasks are used frequently when you create and manage project cables.

Create a Reference Cable

This procedure explains how to create a new cable in the **Reference Data Explorer**. You must first create reference cables with manufacturer specifications to be able to define cables in your project. For more information, see *Create a Reference Cable*, page 57.

Create a Project Cable

This procedure explains how to create a new cable in the **Electrical Index** based on existing reference cable data. For more information, see *Create a Project Cable*, page 182.

Size a Cable

This procedure explains how to size low voltage project power cables. If you associate the cable to a load, the software uses the load data for sizing, or you can size a cable that is not associated with a load by typing in data manually. The algorithm that the software uses for sizing gives a good estimate; however, we recommend that you use specialized cable sizing software for more accurate results. For more information, see *Size a Cable*, page 184.

Batch Size Cables

This procedure explains how to size a group of power cables. For more information, see *Batch Size Cables*, page 186.

Replace Cables

This procedure explains how to replace the reference cable used for a group of project cables. The software replaces the project cables with a standard cable that you previously defined in the **Reference Data Explorer**. For more information, see *Replace Cables*, page 188.

Batch Replace Cable Side Associations

This procedure explains how to replace the cable end connections for a group of non-power cables. It is useful where a large number of cables are connected to the same marshalling rack or cabinet. For more information, see *Batch Replace Cable Side Associations*, page 192.

Associate Cable Sides with Glands

This procedure associates glands with project or reference cables. You can specify different gland types for each cable end. For more information, see *Associate Cable Sides with Glands*, page 193.

Create a Busway

This procedure explains how to create a new busway in the **Electrical Index**. A busway can be used in a similar manner to power cable, other than the ability to perform sizing. You can define the properties of a busway in the **Properties** window. For more information, see *Create a Busway*, page 195.

Create a Project Cable

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Right-click the desired cable folder: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**, and then click **New**.

3. Accept the default item tag that is based on the *naming convention* for the cable, or type your own item tag according to the naming convention.

   💡 **Tips**

   - For more details of naming convention settings, see *Using Naming Conventions*, page 127.

   - If the cable naming convention includes the tag of an associated item and you rename the associated item tag, you must click **Refresh** 🔁 on the toolbar of the **Electrical Index** or the **Electrical Engineer** to update the cable tag.

   - If you are creating a number of cables at one time, you may want to stop at this point and edit the cable properties at a later stage.

4. On the **Cable Common Properties** dialog box, **General** tab, select a cable specification.

5. For the selected specification, do one of the following:

   - If you know the reference cable that you want to use to define your project cable, select that reference cable from the **Reference cable** list. The software populates the properties with data from the reference cable.

   - If you want to perform sizing for a power cable to determine the most suitable reference cable, select an option from the **Conductor arrangement** list.

   💡 **Tip**

   - In addition to sizing a non-sized power cable, you can perform cable sizing when you select a specific reference cable also.

6. Enter other values for the cable properties as required. You must enter values for those properties marked with an asterisk.

7. Click the **Design Data** tab and enter routing and cable length data as required.

📝 **Notes**

- You must associate each power cable in the **Electrical Index** with a reference (catalog) cable. You must therefore first define appropriate reference cables in the **Reference Data Explorer**.

- You access the **Cable Common Properties** dialog box by clicking the **Common Properties** command on the shortcut menu or you can specify in the preferences to open the dialog box directly when you create the cable.

Related Topics
- *Create a Reference Cable*, page 57
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *General Tab (Cable Common Properties Dialog Box)*, page 596
- *Managing Cables Common Tasks*, page 181
- *Size a Cable*, page 184

Edit Project Cable Properties

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand the desired cable folder: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

3. Right-click the desired cable and on the shortcut menu, click **Common Properties**.

4. On the **Cable Common Properties** dialog box, **General** tab, select a cable specification.

5. For the selected specification, do one of the following:

- If you know the reference cable that you want to use to define your project cable, select that reference cable from the **Reference cable** list. The software populates the properties with data from the reference cable (for further details, see Notes at the end of this procedure).

- If you want to select a reference cable by performing and applying sizing for a power cable, enter the necessary data (for details of the properties that are mandatory, see *Required Data for Cable Sizing*, page 538).

💡 **Tip**

- In addition to sizing a non-sized power cable, you can perform cable sizing when you select a specific reference cable also.

6. Enter other values for the cable properties as required. You must enter values for those properties marked with an asterisk.

💡 **Tip**

- If the cable naming convention includes the tag of an associated item and you rename the associated item tag, you must click **Refresh** 🔄 on the toolbar of the **Electrical Index** or the **Electrical Engineer** to update the cable tag.

7. Click the **Design Data** tab and enter routing and cable length data as required.

📝 **Notes**

- When applying sizing for a power cable, suitable reference cables that the software can select from must be defined in the **Reference Data Explorer**.

- When changing the reference cable manually or by applying cable sizing, the software copies all properties relating to cable construction and electrical properties such as ampacity and resistance. The software does not copy project data such as cable tag and description, nor data used for sizing such as estimated and design lengths. The software copies other data such as manufacturer and model only if the value for the project cable is initially empty.

- Whenever the data of the reference cable that a project cable is associated with is changed, the **Compare to Reference** command button in the project cable becomes enabled. This button opens the **Compare Project and Reference Cable Data** dialog box, from which you can compare the current project cable properties with the modified reference cable properties. You can copy the updated reference data to the project cable if required. For a parallel cable assembly, the software applies the reference data to all the cables in the assembly.

- For full details of the most commonly used properties, see the **Cable Common Properties** dialog box. Also, you can edit these properties and additional properties in the **Properties** window or in the Tabular Editor.

Related Topics

Size a Cable

📝 **Note**

- SmartPlant Electrical uses algorithms that give a good estimate for cable sizing; however, we recommend that you use specialized cable sizing software for more accurate results.

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables** > **Power Cables**.

   💡 **Tip**

   - You can also select a cable in the **Electrical Engineer**.

2. Select the desired cable and do one of the following:

   - Click **Edit** > **Common Properties**.

   - Right-click the cable and on the shortcut menu, click **Common Properties**.

   - Press **Ctrl** + **F2**.

3. On the **General** tab of the **Cable Common Properties** dialog box, select the **Sizing required** check box.

4. Select a cable specification.

   💡 **Tip**

   - When sizing the cable, the software selects the appropriate reference cable from the family of cables that you specify under **Cable specification** and with the value that you select under **Conductor arrangement**.

5. Click the **Design Data** tab and enter cable length data.

   💡 **Tip**

   - In the absence of a value for the **Design length** property, the software uses the value in the **Estimated length** box.

6. Click the **Sizing Data** tab and enter data. For full details of the properties for which you must enter values for sizing, see *Required Data for Cable Sizing*, page 538.

   💡 **Tip**

   - If the cable has an associated load, you can copy load data to the cable by clicking **Refresh Data**.

7. Click **Perform Sizing** to perform the sizing calculation.

8. Click **Apply Sizing** to select the recommended reference cable or cables for use as the basis for this cable. The software replaces the currently selected reference cable and updates the cable properties.

   💡 **Tip**

   - If you want to use the shipped reference data provided with the application, please validate that these values are appropriate for cable sizing. Note that shipped reference data is provided to you as uncertified samples only.

9. Click **Apply** to create the sized cable or set of cables in your project.

📝 **Notes**

- For full details of the most commonly used properties, see the **Cable Common Properties** dialog box. Also, you can edit these properties and additional properties in the **Properties** window or in the Tabular Editor.

- If the cable has an attached load, the software copies the values of sizing data properties from the load to the cable.

- If the software cannot find a single reference cable that is suitable for the needed cable size, it selects as many parallel cables as required, up to the maximum allowed, which you can determine in Options Manager. Also in Options Manager, you can select the minimum conductor size that the software looks for when sizing cables.

- You have to resize a power cable if you reassign it to another load or modify the cable properties. Also, we recommend that you generate a report or use the Tabular Editor to view which power cables have their **Cable Sizing Flag** property set to **True** (same as the **Sizing required** check box on the **Cable Common Properties** dialog box. This enables you to see the power cables that need resizing.

Related Topics

Batch Size Cables

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables** > **Power Cables**.

2. Click the **Power Cables** folder and adjust the size of the panes to view the individual cables in the list view pane.

3. If required, change the view in the list view pane by clicking **View Settings** to toggle through each view of the items in the list view pane, or click the arrow beside the icon and select the desired view.

4. Hold **Ctrl** while clicking the cables that you want to size.

5. Click **Actions** > **Cables** > **Batch Size Cables**.

6. In the **Batch Size Cables** window, do one of the following:

   - For each cable that you want to size, select the **Select** check box.

   - Select the **Select all** check box if you intend to size all the cables in the list.

7. Click **Excel Report** to display the information in this window in an Excel file, from where you can format the layout and print the data as required.

   💡 **Tip**

   - If data required for sizing is missing from a cable, the software cannot size the cable. We therefore recommend that you print the contents of the window or view the **Comments** column to ascertain whether data is missing for any of the cables that you intend to size. You should open the **Cable Common Properties** dialog box for each of those cables and complete the data, and then you can size the cables in batch mode.

8. Click **Apply Sizing** to perform the sizing for the selected cables.

   📝 **Note**

   - When the selected set of cables for batch sizing includes parallel cables, only one tag per parallel cable assembly appears in the display; however the software performs the operation for all the parallel cables in each assembly.

   - The cable data in the **Batch Size Cables** window is read-only. To update property values, you need to do so for each cable individually.

Related Topics

Replace Cables

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand one of the cable folders: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

3. Click the folder and adjust the size of the panes to view the individual cables in the list view pane.

4. If required, change the view in the list view pane by clicking **View Settings** to toggle through each view of the items in the list view pane, or click the arrow beside the icon and select the desired view.

5. Hold **Ctrl** while clicking the cables that you want to replace.

6. Click **Actions** > **Cables** > **Replace Cables**.

7. On the **Replace Cables** dialog box, do one of the following:

   - For each cable that you want to replace, select the check box beside the cable name.

   - Select the **Select all** check box if you intend to replace all the cables in the list.

8. Select a reference cable to replace in the selected project cables.

   💡 **Tip**

   - If required, select properties under **Filter criteria** to filter the list of available reference cables.

📝 **Notes**

- The software replaces cables of the same category. For example, you can only replace a power cable with another power cable.

- The software does not allow you to replace cables in batch mode if the cables that you want to replace have already been terminated. You can replace a cable manually in the **Common Properties** dialog box if the reference cable has the same conductor arrangement as the cable you want to replace.

- The software replaces only the values of the following properties with values from the reference cable that you select:

Cable Category

Cable Specification

Basic Ampacity in Air

Basic Ampacity in Ground

Reactance

Resistivity

Actual Conductor Size

Additional Conductor Size

Air Ampacity Temperature

Armor Type

Bending Radius

Conductor Arrangement

Conductor Material

Conductor Size

Inner Insulation Temperature

Insulation Color

Insulation Material

Insulation Temperature rating

Insulation Thickness

Insulation Voltage Rating

Jacket Color

Jacket Insulation Material

Jacket Insulation Temperature

No of Additional Conductors

No of Conductors

No of Conductors in Group

No of Sets

Outside Diameter

Overall Shield

Pull Tension

Weight per Length

Cable Formation

Electrical Equipment Class

Electrical Equipment Sub Class

Electrical Equipment Type

Stores Part Number

- The software clears the values of the following properties:

Calc Starting Voltage Drop

Calc Voltage Drop Percent

- The software retains the values of all other properties of the selected cables.

- When the selected cables for replacing include parallel cables, only one tag per parallel cable assembly appears in the display; however the software performs the operation for all the parallel cables in each assembly. If descriptions exist for the individual parallel cables, the software retains those descriptions.

- If you want to replace a group of parallel cables generated by the sizing procedure with cables having a different formation, for the **Reference cable** property, you must select **Non-sized power cable**. Otherwise, you can only replace parallel cables with another group of cables that have the same formation. For example, you can only replace three parallel cables of 300 mm^2 (3*300 mm^2) with another configuration of 3 parallel cables, such as 3*90 mm^2, 3*(3*25 mm^2+1*10 mm^2), and so forth.

Related Topics

Delete a Cable

1. In the **Reference Data Explorer** or the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand one of the cable folders: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

3. Right-click the cable that you want to delete, and on the shortcut menu, click **Delete**.

4. At the prompt, click **Yes** to confirm the deletion.

Note

- If you select more than one cable for deletion, the software records any deletion failures in the **Batch Deletion** log file. For the procedure for creating and viewing the log file, see *Generate Error Log Files*, page 120.

Deletion Rules for Cables

The following rules apply when you delete a cable in the **Reference Data Explorer** or the **Electrical Index**:

Item	Precondition	Result
Reference cable	There is no project cable that uses the properties of the reference cable.	The software deletes the cable.
Project cable	None.	The software deletes the cable and moves any items in the electrical hierarchy below the deleted cable to the root of the **Electrical Engineer**.
Control station cable	None.	The software deletes the cable.

Related Topics
- *Create a Project Item*, page 53
- *Delete an Item*, page 40
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38
- *Rename an Item*, page 39

Batch Replace Cable Side Associations

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Select the desired cable folder: **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

   💡 **Tip**

   - You cannot perform this operation for power cables.

3. In the lower pane, select the desired cables whose end connections you want to replace.

4. Click **Actions** > **Cables** > **Batch Cable Side and Gland Association**.

5. On the **From** or **To** cable side, clear the **Clear existing connection** check box to allow replacement of existing cable associations for the cables that you select.

6. Beside the check box, click **Find** to open the **Find** dialog box.

7. From the **Equipment type** list, select the desired equipment type.

8. Click **Find Now**.

9. In the **Results** data window, select the item that you want to associate with the cable sides, and click **OK**.

10. From the cable list in the **Batch Cable Side and Gland Association** dialog box, select the check boxes beside the cables for which you want to make cable side associations and click **OK**.

Related Topics
- *Batch Dissociate Cable Sides and Glands*, page 194
- *Managing Cables Common Tasks*, page 181

Create Reference Glands

1. In the **Reference Data Explorer**, under **Wiring Equipment**, right-click the **Glands** folder.

2. Click **New Gland**.

3. On the **Reference Gland Common Properties** dialog box, **General** tab, type values for each property as desired.

4. Click the **Details** tab.

5. Enter values for the properties as required. You must enter values for those properties marked with an asterisk.

Related Topics
- *Associate Cable Sides with Glands*, page 193
- *Reference Gland Common Properties Dialog Box*, page 620

Associate Cable Sides with Glands

1. In the **Electrical Index**, select a cable and open its **Cable Common Properties** dialog box.

   ### 💡 Tip

   - You can associate glands with a reference cable if you want to create project cables from it with glands already assigned.

2. Click the **Connections** tab.

3. On the **From** or **To** cable side, beside the appropriate **Gland** property, click **Find**.

   ### 📝 Note

   - After you have associated glands with a number for cables, you can view the bill of materials for the glands by generating a Gland Material Take-Off report.

   Related Topics
   - *Create Reference Glands*, page 193
   - *Managing Cables Common Tasks*, page 181

Batch Associate Cable Sides with Glands

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Select the desired cable folder: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

3. In the lower pane, select the desired cables for which you want to associate glands.

   ### 💡 Tip

   - You can also perform the cable selection in the Tabular Editor.

4. Click **Actions** > **Cables** > **Batch Cable Side and Gland Association**.

5. On the **From** or **To** cable side, clear the **Clear existing gland** check box to allow replacement of existing cable glands for the cables that you select.

6. Beside the check box, click **Find** to open the **Find** dialog box.

7. Click **Find Now**.

8. In the **Results** data window, select the gland that you want to associate with the cable sides, and click **OK**.

9. From the cable list in the **Batch Cable Side and Gland Association** dialog box, select the check boxes beside the cables for which you want to make gland associations and click **OK**.

Batch Dissociate Cable Sides and Glands

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Select the desired cable folder: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

   💡 **Tip**

   - For power cables, you can dissociate glands only.

3. In the lower pane, select the desired cables.

4. Click **Actions** > **Cables** > **Batch Cable Side and Gland Association**.

5. On the **From** or **To** cable side, select the **Clear existing connection** check box to dissociate cable side associations for the cables that you select.

6. On the **From** or **To** cable side, select the **Clear existing gland** check box to dissociate cable glands for the cables that you select.

7. From the cable list, select the check boxes beside the cables for which you want to dissociate the items and click **OK**.

Related Topics
- *Associate Cable Sides with Glands*, page 193
- *Batch Associate Cable Sides with Glands*, page 194
- *Batch Replace Cable Side Associations*, page 192
- *Managing Cables Common Tasks*, page 181

Create a Busway

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Busways**.

2. Right-click the folder and click **New Busway**.

3. Rename the busway if you require.

4. In the **Properties** window, enter values for the desired busway properties.

Related Topics
- *Managing Cables Common Tasks*, page 181

Cable Routing: An Overview

The cable routing feature allows you to set up a model of the cable routing in your plant. First, you need to create reference cableway components. These are catalog components for which you can specify manufacturer's data. The software allows you to specify one of three possible categories of reference cableway components: trays, conduits, or miscellaneous. You can make space reservations by defining cableways and segments (the physical conduits) in your plant, and later associate the segments with reference components. Alternatively, you can specify the reference components for the segments at the time when you create them. After you define the cableways and segments in your plant, you can assign cables to specific routing paths, singly or in batch mode.

The software allows you to create predefined routing paths as a way of specifying physical relationships between cableways. You can specify the cableways in a routing path first, and then later define the segments.

You must define a segregation level for the cables you intend to route, as well as for the cableway segments. Assignment of segregation levels allows you to specify separate groups of cables that you would not normally route together along the same path, for example, high and low voltage cables.

Related Topics
- *Batch Route Cables*, page 199
- *Define Routing for a Single Cable*, page 199
- *Managing Cable Routing Common Tasks*, page 196

Managing Cable Routing Common Tasks

The following tasks are used frequently in cable routing.

Create a Cableway and Segments

This procedure explains how to create a cableway and its segments in the **Electrical Index**. In order to define a routing path to which you can assign cables, you must first create cableways and their segments. Initially, these cableways and segments can specify space reservations. For more information, see *Create a Cableway and Segments*, page 197.

Assign Cableway Components to Segments

After you create a cableway, use this procedure to define and edit essential cableway properties. For more information, see *Assign Cableway Components to Segments*, page 198.

Define Routing for a Single Cable

This procedure explains how to define a routing path for a single cable. For more information, see *Define Routing for a Single Cable*, page 199.

Batch Route Cables

This procedure explains how to define routing paths for cables in batch mode. Batch cable routing is useful when you want to assign multiple cables belonging to the same segregation level to a routing path. For more information, see *Batch Route Cables*, page 199.

Edit Cableway Properties

This procedure explains how to modify cableway properties, including inserting, deleting, and moving segments. For more information, see *Edit Cableway Properties*, page 200.

Set The Method For Segment Fill Validation

This procedure explains how to set the desired segment fill validation method. For more information, see *Set The Method For Segment Fill Validation*, page 203.

Set the Segment Fill Calculation Criterion

This procedure explains how to set the segment fill calculation criterion for segment fill validation. Note that when using the NEC validation method, the **Calculation criteria** list is not available. For details, see *Set the Segment Fill Calculation Criterion*, page 206.

Shipped Reports

A number of reports that come shipped with the software are available to assist you in managing your cable routing efficiently. These include:

- Segment Width Estimate
- Cableway Component Material Take-Off
- Cableway Segment Schedule
- Segment Fill

For more information, see *Shipped Reports*, page 390.

Related Topics
- *Cable Routing: An Overview*, page 195
- *Clear Cable Routing*, page 201
- *Define Routing for a Single Cable*, page 199

Create a Cableway and Segments

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cableways**.
2. Right-click the **Cableways** folder, and on the shortcut menu click **New Cableway**.
3. For each routing segment that you need to create in the cableway, right-click the cableway, and on the shortcut menu click **New Cableway Segment**.
4. For each routing segment that you created, select the segment, and in the **Properties** window, define the segment length and other properties.

   💡 **Tip**

   - You can select a *segregation level* only if you have not assigned any cables to the segment.
5. Right-click the cableway, and on the shortcut menu, click **Common Properties**.
6. On the **Cableway Common Properties** dialog box, sort the segments in the desired sequence by selecting segments and clicking **Up** or **Down**.

Related Topics
- *Batch Route Cables*, page 199
- *Cable Routing: An Overview*, page 195
- *Define Routing for a Single Cable*, page 199
- *Edit Cableway Properties*, page 200
- *Managing Cable Routing Common Tasks*, page 196

Assign Cableway Components to Segments

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cableways**.

2. Expand the desired cableway, and select the desired segment.

3. Right-click the segment, and on the shortcut menu, click **Common Properties**.

4. On the **Cableway Segment Common Properties** dialog box, under **Reference component**, click **Find**.

   💡 **Tip**

   - To filter the selection for a particular reference component category, under **Category**, select **Tray**, **Conduit**, or **Miscellaneous** before you click **Find**.

5. On the **Find** dialog box, select the desired reference component.

   💡 **Tip**

   - The software fills values for certain properties from the reference component, if available.

Related Topics
- *Batch Route Cables*, page 199
- *Cable Routing: An Overview*, page 195
- *Create a Cableway and Segments*, page 197
- *Define Routing for a Single Cable*, page 199

Define Routing for a Single Cable

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**, and the desired folder among the following: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

2. Right-click the cable for which you want to define routing, and on the shortcut menu click **Common Properties**.

3. In the **Cable routing** group box, for the first segment, select the cableway and segment from the relevant lists.

4. For each succeeding segment, click **Add**, and then select the cableway and segment from the relevant lists.

Related Topics
- *Batch Route Cables*, page 199
- *Cable Routing: An Overview*, page 195
- *Managing Cable Routing Common Tasks*, page 196

Batch Route Cables

Important

- The software does not verify routing compatibility of cables with varying voltage levels; therefore you must make sure that your practices permit the cable combinations that you define for a given routing path. If desired, you can use the Rule Manager to define criteria for the type of cables that you allow to belong to the same segregation level.

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**, and select the desired folder (**Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**).

2. In the list view pane, press **Ctrl** while selecting the cables that you want to route.

   Tips

   - All the cables that you select must have the same *segregation level and you must define the actual cable length (estimated length, design length, or routing length) for each cable that you add to the routing*.

   - When routing cables of more than one category, create a Tabular Editor view for the desired cables and select from the view the cables that you want to route.

3. Click **Actions** > **Cables** > **Batch Route Cables**.

4. On the **Batch Route Cables** dialog box, make sure that the **Clear routing from selected cables** check box is cleared.

5. In the **Cable routing** group box, for the first segment, select the cableway and segment from the relevant lists.

6. For each succeeding segment, click **Add**, and then select the cableway and segment from the relevant lists.

7. In the lower data window, do one of the following:

   - For each cable that you want to assign to the routing path, select the check box beside the cable tag.

   - To assign all of the displayed cables, select the **Select all** check box.

Caution

- The new cable routing overwrites any existing values under **Routing Path**.

Note

- For a parallel cable assembly, when you define a cable routing for one of the cables, the software automatically copies the routing to all of the cables in the assembly.

Edit Cableway Properties

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cableways**.

2. Right-click the desired cableway, and on the shortcut menu click **Common Properties**.

3. If desired, modify the cableway tag and description.

4. Under the **Segments** section, modify the segments by doing any of the following:

   - Click **Add** to insert a new segment at the end of the list.

   - Click **Insert** to insert a new segment above the currently selected segment.

   - Click **Remove** to remove the currently selected segment from the cableway.

   - Sort the segments in the desired sequence by selecting segments and clicking **Up** or **Down**.

   💡 **Tip**

   - If you have assigned cables to the segment, you cannot remove or change the sequence of the existing segments; you can only insert new segments by clicking **Add** or **Insert**. When you insert a new segment and define its properties, the software automatically updates the routing for the assigned cables.

Related Topics

Clear Cable Routing

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**, and select the desired folder (**Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**).

2. In the list view pane, press **Ctrl** while selecting the cables from which you want to clear routing.

   💡 **Tip**

   - All the cables that you select must have the same segregation level.

3. Click **Actions** > **Cables** > **Batch Route Cables**.

4. On the **Batch Route Cables** dialog box, select **Clear routing from selected cables**.

5. In the lower data window, do one of the following:

   - For each cable from which you want to clear routing, select the check box beside the cable tag.

   - To clear routing for all of the displayed cables, select the **Select all** check box.

Related Topics
- *Cable Routing: An Overview*, page 195
- *Managing Cable Routing Common Tasks*, page 196

Segment Fill Validation: An Overview

You can perform segment fill validation using one of the two methods, the Conventional or the National Electric Code (NEC) method. Note that the software can calculate the fill percentage only if the segment is associated with a reference raceway component.

Conventional Method

The Conventional method is based on the physical properties of raceway components. This method is commonly used and it has a less restrictive approach than the NEC method. For details on how to set the desired method (Conventional or NEC), see *Set The Method For Segment Fill Validation*, page 203

Tray Fill Calculation

The software calculates tray segment fill according to the following criteria:

- When using the Conventional method, the segment fill calculation is based on the type of reference tray that you select on the **Reference Cableway Component Common Properties** dialog box. The following types of reference trays are available:

- Covered tray segment - Applies to trays and miscellaneous cableway components. The software calculates the total area occupied by the cables assigned to the segment. The software validates this value by ensuring that it does not exceed the allowed area (the product of tray width, tray depth and Fill factor), SmartPlant Electrical validates the covered tray fill segment calculation by area. Furthermore, the software ensures that the outer cable diameter of any cable does not exceed the tray depth. Note that for covered trays the fill factor value can not exceed the value of 1.

- Uncovered tray segment - Applies to trays and miscellaneous cableway components. The software calculates the sum of the outer diameters of the cables assigned to the segment. SmartPlant Electrical validates this value by ensuring that it does not exceed the product of the segment width, the fill factor, and the number of layers. The software validates the uncovered tray fill segment calculation by diameter.

- The segment fill calculation is based on tray width or tray area. You select the desired criterion from the **Calculation criteria** list on the **Cableway Segment Common Properties** dialog box. When selecting the **Auto** option from the **Calculation criteria** list, SmartPlant Electrical calculates the segment fill automatically. The software performs the calculation according to the type of reference tray that you selected on the **Reference Cableway Component Common Properties** dialog box. Selecting the **Covered** check box instructs the software to calculate the segment fill according to the **Area** criterion. If you do not select this check box, SmartPlant Electrical performs the calculation according to **Width**.

📝 **Note**

- For details on how to Set the segment fill calculation criterion, see *Set the Segment Fill Calculation Criterion*, page 206.

Conduit Fill Calculation

The Conduit fill calculation applies to conduits. The software calculates the total area occupied by the cables assigned to the segment. SmartPlant Electrical validates this value by ensuring that it does not exceed a quarter of the allowed area of the conduits. The software calculates the allowed area of the conduit starting from the conduit diameter and the fill factor. The minimum allowed conduit diameter is based on the following formula:

$$D \geq 2 * (\sqrt{\sum d_i^2}) * f$$

Where:

- D = Minimum inner conduit diameter
- di = Individual cable outer diameter
- f = Fill factor

📝 **Notes**

- For conduits, the fill factor value can not exceed the value of 1.

- You need to define cable diameter and weight per unit length for each associated reference cable by entering the desired values in the **Reference Cableway Component Common Properties** dialog box.

- When performing validations based on cable diameter, the software assumes that the cables are laid side-by-side. Therefore the software does not allow for cables that can pass along the interstices between larger cables.

NEC Validation Method

The NEC method is based on given space factors. Unlike the Conventional tray segment validations, the NEC method does not allow flexibility like the Conventional method regarding tray segment fill validation. NEC dictates the fill requirements for various cases of installations, such as cable formations, cable sizes, insulation voltages and tray widths. The calculation applies to the cables routed on the segment as part of the routing. Note that the **Fill factor** values can only be between 0 and 1, as defined by the NEC code. The software takes into account the fill factor that you specify and the maximum weight per length unit allowed on tray. If any of the actual fill values exceed the allowed values, the software rejects them from being routed and displays a warning message on the **Cableway Segment Common Properties** box.

Conduit Fill Factor

When using the NEC validation method in a conduit, the software sets the fill factor according to the number of cables in the conduit:

- If the number of cables in a conduit = 1, the fill factor = 0.53

- If the number of cables in a conduit = 2, the fill factor = 0.31

- If the number of cables in a conduit > 2, the fill factor = 0.4

Note

- You cannot modify these additional fill factors.

The NEC rules for tray segment fill validation are based on the following primary parameters and their various combinations:

- The insulation voltage rating of the cable that it is carrying. The NEC method divides the voltage into two groups, above 2000v and 2000v, and below.

- Cables arrangement (single core or multi core). The code does not address a combination of these cable categories, therefore SmartPlant Electrical does not support such a combination.

- A combination of power and non-power carrying cables (instrumentation, control, grounding).

- Type of tray on which cables are laid; whether it is **Ladder**, **Ventilated trough**, **Solid bottom tray**, **Ventilated channel**, **Solid channel.**

- Conductor sizes

- Tray dimensions

For each of the possible combinations of the above mentioned parameters, NEC method specifies (NEC 2005 handbook) one or more of the following criteria:

- The allowable number of layers
- The allowable fill area
- The allowable tray width percentage

All the limiting numbers of the allowable width and cross sectional areas are automatically set in SmartPlant Electrical and cannot be changed.

SmartPlant Electrical generates a NEC tray fill validation log file that documents tray validations. For details on how to generate a log file, see *Generate Error Log Files*, page 120. In the generated log file you can view the properties of the specific tray fill validation, such as - Item type, date of validation, Layout ID, and more. The NEC tray fill validation log file is based on SmartPlant Electrical Tray Fill Calculation Logic, which is based on: NFPA 70 National Electrical Code 2005 Edition.

For more details see, ??SmartPlant Electrical Cable Tray Fill Calculation Logic??. For the NEC tables refer to article 392, section 9 and 10 in the NEC 2005 Edition.

Related Topics
- *General Tab (Cableway Segment Common Properties Dialog Box)*, page 632
- *General Tab (Reference Cableway Component Common Properties Dialog Box)*, page 636
- *Managing Cable Routing Common Tasks*, page 196

Set the Method For Segment Fill Validation

1. Open the Option Manager.
2. Click **Options** > **General Settings**.
3. On the **General Settings** dialog box, in the **Cable fill validation to follow NEC regulations** pane, select from the list one of the following:

   - Select **No** (default) if you want to apply the Conventional method.
   - Select **Yes** if you want to apply the NEC (national Electric Code) method.

4. Click ![save] to save.

Related Topics
- *General Tab (Cableway Segment Common Properties Dialog Box)*, page 632
- *General Tab (Reference Cableway Component Common Properties Dialog Box)*, page 636
- *Segment Fill Validation: An Overview*, page 202

Set the Segment Fill Calculation Criterion

1. In the Option Manager, set the segment fill validation to **Conventional**, for more details see *Set The Method For Segment Fill Validation*, page 203.

2. In SmartPlant Electrical, on the **Electrical Index** tree-view, expand the **Wiring Equipment** folder.

3. Expand the **Cableways** folder and select the desired cableway or create a new cableway and segments, for details see *Create a Cableway and Segments*, page 197.

4. Right click a desired segment, on the shortcut menu, click **Common Properties**.

5. On the **Cableway Segment Common Properties** dialog box, from the **Calculation criteria** list, select the desired calculation criterion (**Auto**, **Area**, or **Width**).

   - **Auto** — When selecting the **Auto** option, the calculation is preformed automatically according to the type of reference tray that you selected on the **Reference Cableway Component Common Properties** dialog box.

   - **Area**— The software calculates the segment fill by area.

   - **Width**— The software calculates the segment fill by width.

Related Topics
   - *General Tab (Cableway Segment Common Properties Dialog Box)*, page 632
   - *General Tab (Reference Cableway Component Common Properties Dialog Box)*, page 636
   - *Segment Fill Validation: An Overview*, page 202

Cable Drum Assignment: An Overview

The cable drum assignment feature helps you plan cabling tasks and optimize cable drum use. It also allows you to determine cable lengths so that you can purchase cables while minimizing a shortage or surplus of cable at the end of the project.

✎ Note

- Before you begin, make sure that you have defined cable pulling areas in the Data Dictionary Manager. For more information, see Data Dictionary Manager Online Help.

Although the order may vary, the following activities are essential elements in cable drum assignment for a given pulling area:

- Creating reference cables

- Creating project cables based on the reference cables

- Creating cable drums for all of the reference cables used in the pulling area

- Assigning all of the project cables in the pulling area to cable drums

For details of the formula used for the **Cable Drum Assignment Wizard**, see the Cable Algorithms.xls Excel file.

(If the **Cable Algorithms.xls** file does not display when you click the link, you can open the file manually from the folder where the SmartPlant Electrical program is installed.)

Related Topics
- *Assign Multiple Cables to a Cable Drum*, page 210
- *Cable Drum Common Properties Dialog Box*, page 613
- *Create a Cable Drum*, page 209
- *Managing Cable Drums Common Tasks*, page 208

Managing Cable Drums Common Tasks

The following tasks are used frequently when you create and manage cable drums.

Create a Cable Drum

This procedure explains how to create a new cable drum in the **Electrical Index**. In order to assign cables to cable drums in your project, you must first create cable drums for each reference cable in your project. For more information, see *Create a Cable Drum*, page 209.

Edit Cable Drum Properties

After you create a cable drum, use this procedure to select a drum reference cable, and to define and edit other cable drum properties. For more information, see *Edit Cable Drum Properties*, page 210.

Assign Multiple Cables to a Cable Drum

This procedure explains how to assign one or more cables to a cable drum that you created in the **Electrical Index**, and for which you set a reference cable. For more information, see *Assign Multiple Cables to a Cable Drum*, page 210.

Assign a Single Cable to a Cable Drum

This procedure explains how to assign a cable that you are editing to a given cable drum. For more information, see *Assign a Single Cable to a Cable Drum*, page 211.

Batch Assign Cables to Cable Drums

This procedure explains how to assign cables to multiple drums using the **Assign Drums to Cables Wizard**. For more information, see *Batch Assign Cables to Cable Drums*, page 213.

Unassign Cables from a Cable Drum

This procedure explains how to remove cables that are assigned to a drum. For more information, see *Unassign Cables from a Cable Drum*, page 213.

Related Topics
- *Cable Drum Assignment: An Overview*, page 207

Create a Cable Drum

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Drums**.

2. Right-click the **Drums** folder, and on the shortcut menu click **New Drum**.

3. Right-click the drum that you created, and on the shortcut menu click **Common Properties**.

4. On the **Cable Drum Common Properties** dialog box, under **Drum Tag**, edit the tag name as necessary.

5. To set the required drum reference cable, beside **Reference cable** click ⬚, and on the **Select Reference Cable** dialog box, select the desired reference cable, and click **OK**.

6. Enter values for the other cable drum properties. For details, see *Cable Drum Common Properties Dialog Box*, page 613.

📝 **Notes**

- For calculation details, see *Rules Governing Cable Drum Calculations*, page 212.

Edit Cable Drum Properties

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Drums**.

2. Right-click the desired cable drum, and on the shortcut menu click **Common Properties**.

3. On the **Cable Drum Common Properties** dialog box, under **Drum Tag**, edit the tag name as necessary.

4. To set the required drum reference cable, beside **Reference cable** click ⬚, and on the **Select Reference Cable** dialog box, select the desired reference cable, and click **OK**.

5. Enter values for the other cable drum properties. For details, see *Cable Drum Common Properties Dialog Box*, page 613.

📝 **Notes**

- You cannot change the drum reference cable when cables are assigned to the drum.

- For calculation details, see *Rules Governing Cable Drum Calculations*, page 212.

Assign Multiple Cables to a Cable Drum

1. Open the **Cable Drum Common Properties** dialog box for the cable drum to which you want to assign cables, and edit the values as needed. For details, see *Edit Cable Drum Properties*, page 210.

   💡 **Tip**

   - Ensure that you have specified a reference cable for the cable drum and a pulling area for both the cable drum and the cables. You must also enter values for **Minimum cable length to order** and **Maximum drum capacity** - you can enter these values in the Data Dictionary, the Options Manager project-wide parameters, or in the reference cable properties.

2. Click **Add**.

3. On the **Find** dialog box, click **Find Now**.

4. In the **Results** data window, select the cables that you want to assign to the current drum, and click **OK**.

📝 **Notes**

- You can only assign to a particular drum cables that are associated with the designated reference cable and that belong to the same pulling area as the drum.

- For calculation details, see *Rules Governing Cable Drum Calculations*, page 212.

Related Topics
- *Assign a Single Cable to a Cable Drum*, page 211
- *Batch Assign Cables to Cable Drums*, page 213
- *Cable Drum Common Properties Dialog Box*, page 613
- *Find Dialog Box*, page 580
- *Managing Cable Drums Common Tasks*, page 208
- *Unassign Cables from a Cable Drum*, page 213

Assign a Single Cable to a Cable Drum

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand the desired folder among the following: **Power Cables**, **Control Cables**, **Grounding Cables**, or **Instrumentation Cables**.

3. Right-click the cable that you want to assign, and on the shortcut menu click **Common Properties**.

4. On the **Cable Common Properties** dialog box, on the **Design Data** tab, click **Find**.

5. On the **Find** dialog box, click **Find Now**.

6. In the **Results** data window, select the cable drum to which you want to assign the cable, and click **OK**.

Note

- If a cable exceeds the maximum cable drum capacity, the software automatically defines the required cable splices for that cable. You can view the specified lengths of the cable cuts on the **Design Data** tab of the **Cable Common Properties** dialog box. Click **View** in the **Drum assignment** group box.

- For calculation details, see *Rules Governing Cable Drum Calculations*, page 212.

Related Topics
- *Assign Multiple Cables to a Cable Drum*, page 210
- *Cable Drum Common Properties Dialog Box*, page 613
- *Find Dialog Box*, page 580
- *Managing Cable Drums Common Tasks*, page 208
- *Unassign Cables from a Cable Drum*, page 213

Rules Governing Cable Drum Calculations

When you assign cables to cable drums, there are specific rules that govern how the software determines whether it is possible for you to assign a given cable to a drum. These rules apply whether you make the assignment from the **Cable Drum Common Properties** dialog box or from the **Cable Common Properties** dialog box.

The following rules apply:

- For each cable, SmartPlant Electrical uses the *actual length* value.

- The value under **Total actual length assigned** is the sum of the actual lengths of all the assigned cables.

- If assigning a given cable would cause the **Total actual length assigned** value to be greater than the value that you set under **Maximum cable length**, the software does not allow the cable assignment.

- You have the option to make allowance for cable cutting error by entering a value under **Drum spare percent**. The software includes this percentage in the **Total actual length assigned** value.

Related Topics
- *Assign a Single Cable to a Cable Drum*, page 211
- *Assign Multiple Cables to a Cable Drum*, page 210
- *Cable Drum Common Properties Dialog Box*, page 613
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597

Batch Assign Cables to Cable Drums

1. Click **Actions** > **Cables** > **Assign Drums to Cables Wizard** to open the **Assign Drums to Cables Wizard**.

2. On the **Welcome** page, click **Next**.

3. On the **Drum and Cable Options** page, select the check boxes as you require for the following options:

   - Include cables already assigned to drums.

   - Create new drums if needed.

   - Delete empty drums.

4. Do one of the following:

   - Select a single reference cable - beside **Reference cable**, click , to open the **Select Reference Cable** dialog box.

   - Select the **Select all reference cables** check box.

5. Do one of the following:

   - From the **Pulling area** list, select a single pulling area.

   - Select the **Select all pulling areas** check box.

6. Click **Next**.

7. On the **Select Cables** page, do one of the following:

   - For each available cable that you want to assign to a drum, select the **Select** check box beside the cable tag.

   - To select all of the cables that are available for assignment, select the **Select all** check box.

   💡 **Tips**

   - You can click the icon at the top right of the page [icon] to display the page in full-screen mode. You must click the icon again to restore the page to normal view before you can proceed to the next step.

   - You cannot select cables for which pulling area or cable length values are missing. To update the cable data, highlight the cable and click **Common Properties**.

   - If you select any cables that have restricted access rights, the software does not assign them to drums, and the names of those cables appear in the error log.

8. Click **Next**.

9. On the **Drum - Cable Assignment Summary** page, view the details of the cables you have selected to assign to cable drums.

10. Click **Next**.

11. On the **Completing the Assign Drums to Cables Wizard** page, view summary data for the cable - drum assignment and click **Finish** to close the wizard.

📝 **Note**

   - If a cable exceeds the maximum cable drum capacity, the software automatically defines the required cable splices for that cable. The **Assign Drums to Cables Wizard** specifies the lengths of the cable cuts on the **Completing the Assign Drums to Cables Wizard** page.

Related Topics

Unassign Cables from a Cable Drum

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Drums**.

2. Right-click the cable drum from which you want to unassign cables, and on the shortcut menu click **Common Properties**.

3. In the **Assigned cables** data window, select the desired cable and click **Remove**.

Note

- You can also unassign a cable from a cable drum on the **Design Data** tab of the **Cable Common Properties** dialog box.

Related Topics
- *Assign a Single Cable to a Cable Drum*, page 211
- *Assign Multiple Cables to a Cable Drum*, page 210
- *Cable Drum Common Properties Dialog Box*, page 613
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *Managing Cable Drums Common Tasks*, page 208

Associating Electrical Equipment Common Tasks

The following tasks are used frequently when you associate electrical equipment. There are several ways that you can make electrical associations. The particular method that you choose depends on your workflow and specific requirements.

Associate Electrical Equipment with Feeders

This procedure shows how to associate an electrical item with a feeder using the **Electrical Motor Common Properties**, **Static Load Common Properties**, or **Converting Equipment Common Properties**, dialog box. On the appropriate dialog box, the **Feeder Data** tab options allow you to associate the selected electrical item with a power distribution board, bus, cell, and circuit as required. For more information, see *Associate Electrical Equipment with Feeders*, page 216.

Associate Loads with a Power Distribution Board in Batch Mode

This procedure explains how to associate a number of *loads* with a selected power distribution board. The software makes the associations and then sums up the electrical consumption and the full load current that the associated loads require. This calculation facilitates the power distribution board design at an early planning stage. Note that you can choose one of the following calculation modes:

- Calculate the total consumed power only for those loads that are connected directly to the current bus.

- Calculate the total consumed power for all the loads fed by the current bus as well as the loads that are fed by the buses that are connected to the current bus (as specified in Options Manager).

For more information, see *Associate Loads with a PDB in Batch Mode*, page 219.

Associate a Single Load with a Power Distribution Board or a Bus

This procedure explains how to associate a single *load* with a selected power distribution board or bus. The software makes the associations and sums up the electrical consumption and the full load current that the associated load requires. This calculation facilitates the power distribution board design at the early planning stage. For more information, see *Associate a Single Load with a PDB*, page 221.

Associate Bus Risers with Couplers

This procedure shows how to associate *bus riser circuits* with *coupler circuits* to create a bus tie. For more information, see *Associate Bus Riser Circuits with Coupler Circuits*, page 226.

Associate a Control Station with an Item

This procedure shows how to associate a control station with an electrical item. Note that this association does not appear in the **Electrical Engineer**. For more information, see *Associate a Control Station with an Item*, page 226.

Associate a Control Station with Cables

This procedure explains how to associate a control station with cables. For more information, see *Associate a Control Station with Cables*, page 227.

Associate Control Station Cables with Equipment Circuits

This procedure explains how to associate manually instrumentation or control cables of control stations with the circuit belonging to the equipment that the control station operates or monitors. You can also specify automatic association of control station cables with equipment circuits from the Options Manager. For more information, see *Associate Control Station Cables with Equipment Circuits*, page 227.

Associate a Power Cable with Electrical Equipment

This procedure explains how to associate a power cable with an electrical item by making the association in the **Electrical Engineer**. For more information, see *Associate a Power Cable with Electrical Equipment*, page 228.

Associate Additional Power Cables with Electrical Equipment

This procedure explains how to add a power cable to an electrical item in parallel to an existing power cable in the **Electrical Engineer**. For more information, see *Associate Additional Power Cables with Electrical Equipment*, page 229.

Associate a Non-Power Cable with Electrical Equipment

This procedure explains how to associate a control, grounding, or instrumentation cable with an electrical item using the **Cable** form. For more information, see *Associate a Non-Power Cable with Electrical Equipment*, page 230.

Create a Daisy-Chain Connection Pattern

This topic provides the rules for connecting electrical items in a daisy-chain pattern. For more information, see *Create a Daisy-Chain Connection Pattern*, page 231.

View Associated Drawings of an Item

This procedure allows you to select an item from the **Electrical Index** and view a list of all the drawings that the item appears on. You can filter the results by drawing type and then open a specific drawing. For more information, see *View Associated Drawings of an Item*, page 372.

Associate Electrical Equipment with Feeders

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Loads** > **Motors** or **Static Electrical Equipment** or **Miscellaneous Electrical Equipment** and select an electrical equipment item.

   💡 **Tips**

   - You can only associate equipment that is not already assigned to an existing circuit. To verify whether an association exists, right-click the item and click **Find in Electrical Engineer**.

   - You can also associate a load with a feeder by dragging the load to the appropriate feeder circuit in the **Electrical Engineer**. For details of this procedure, see *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81.

2. Right-click the selected electrical item and click **Common Properties** to open the appropriate dialog box.

3. Click the **Feeder Data** tab.

4. In the **Location** group box, under **Power distribution board**, select a power distribution board.

   💡 **Tip**

   - If you select a PDB without specifying a bus, on clicking **Apply** or **OK**, the software creates the new electrical item without a circuit.

5. Under **Bus**, select a bus with which you want to associate the current electrical item.

   💡 **Tip**

   - If required, you can select a cell in the **Cell** list to filter the circuits located in that cell.

6. Do one of the following to associate the new item with a feeder circuit:

   - Under **Circuit**, select an existing circuit with which you want to associate the current electrical item.

   - Select the **Create circuit** check box to automatically create a new feeder circuit based on the last applied typical circuit. The software creates an empty circuit without any circuit components if the current electrical item is not associated with a typical circuit. If you do not select this check box, the software only associates the selected bus with the loads without creating any circuits. Where a last applied typical circuit is indicated, the software creates the new circuit based on the typical circuit. Also, you can select the **Create cell** check box to automatically create a cell when the software creates the new feeder circuit.

Tips

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply** or **OK**.

7. Continue defining the feeder data as required and click **OK** when done.

Notes

- If you choose to automatically create a feeder circuit with a cell, make sure that in the **Properties** window for the new cell, you select the **In Use** setting for the **Cell Usage** property.

- After specifying a circuit, applying the change, and exiting the dialog box, you can no longer change the feeder data values from the dialog box.

- A circuit can feed more than one electrical item.

- It is possible to connect a number of heat tracers and Miscellaneous Electrical Equipment items in a daisy-chain pattern.

Related Topics

Automatic Creation of Feeder Circuits

The software automatically creates a feeder circuit when you associate a load or any equipment item with a bus. This automatic creation of feeder circuits occurs when you do one of the following:

- Drag a load or an equipment item from the **Electrical Index** to the **Electrical Engineer**.

- Drag a load or an equipment item from one bus to another in the **Electrical Engineer**.

- Select the **Create circuits** option on the **Batch Load Association** dialog box.

- Select the **Create circuit** option on the **Feeder Data** tab of the **Common Properties** dialog box for any motor, static load, converting equipment, and other equipment.

- Select a PDB, bus, and cell without selecting an existing circuit on the **Feeder Data** tab of the **Common Properties** dialog box for any motor, static load, converting equipment, and other equipment.

Whenever you associate a load or an equipment item with a bus for the first time by dragging an item from the **Electrical Index** to the **Electrical Engineer**, the software automatically creates a feeder circuit for the dragged item. Also, whenever you change an existing association in the **Electrical Engineer**, the software automatically creates a new feeder circuit and circuit components on the bus to which you dragged the load or equipment item. In this case, the old feeder circuit that belonged to the dissociated item remains in its original place. You can delete it if you want to or leave it as a spare feeder circuit to associate with another equipment item.

Regardless of whether you drag a load to a bus from the **Electrical Index** to the **Electrical Engineer** or within the **Electrical Engineer**, the software also creates circuit components for the new feeder. In this case, must ensure that you define the cell properties correctly. In the **Properties** window for a cell, select the **In Use** setting for the **Cell Usage** property. When dissociating the last load from a circuit, select the **Installed Spare** setting for the **Cell Usage** property.

If the load that you drag is associated with a typical circuit, the software creates the necessary circuit components as they are defined in the typical circuit. If the item is not associated with a typical circuit, the software creates a feeder circuit without any circuit components. If the item that you drag is not a load, the software creates a feeder circuit without any circuit components.

Also, you can instruct the software to automatically create a feeder circuit and a cell when you set the **Feeder** options on the **Common Properties** dialog box of any load or equipment item as well as when associating loads with buses in batch mode. For more information about batch load association, see *Associate Loads with a PDB in Batch Mode*, page 219.

Related Topics
- *Associate Electrical Equipment with Feeders*, page 216
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Associating Electrical Equipment Common Tasks*, page 215
- *Change an Existing Electrical Association*, page 85
- *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81
- *Working with the Electrical Engineer: An Overview*, page 46

Associate Loads with a PDB in Batch Mode

1. In the **Electrical Index**, expand the **Electrical Equipment** folder hierarchy and navigate to the desired folder, such as **Motors**, **Static Electrical Equipment**, **Converting Equipment**, and so forth.

2. In the list view pane, select the items that you require.

   💡 **Tip**

   - You can also associate a single load without selecting a batch of loads in the **Item Tag** pane of the **Electrical Index**. This way you can associate different loads belonging to any load category in the **Electrical Index**. For more details, see *Associate a Single Load with a PDB*, page 221.

   - Also, you can select the desired loads in the Tabular Editor.

3. Click **Actions** > **Batch Load Association**.

4. On the **Batch Load Association** dialog box, under **Power distribution board**, select the desired power distribution board.

5. From the **Bus** list, select a bus with which you want to associate the loads.

6. On the **Bus** tab, specify the rated data for the selected bus as needed and click **Apply**.

7. Click the **Loads** tab.

8. Click **Add Selected Loads** to add the items to the **Associated loads** list. (You selected these items in the list view pane of the **Electrical Index**.)

9. In the **Associated loads** box, scroll to the right and define the phase association of the current load by selecting the appropriate **Phase** check boxes.

   💡 **Tip**

   - For a single-phase load, select one of the **Phase** check boxes. For a three-phase load, select all three **Phase** check boxes.

10. Under **Create circuits and cells**, select the **Create circuits** check box to automatically create a circuit for each associated load that has not been assigned to a circuit. Note that this option does not affect the loads that have already been assigned to circuits.

    💡 **Tips**

    - If you do not select the check boxes under **Create circuits and cells**, the software associates the selected loads with a power distribution board or a bus. You can create circuits at a later time.

- Note that circuits have a property called **CircuitMode** with values **Connected** and **Disconnected**. If you set this property to **Disconnected**, the drill down algorithm will stop the calculation there and will not roll up the loads connected to this feeder.

11. Select the **Create cell** check box to automatically create a cell where the new feeder circuit for the associated load will be created.

💡 **Tip**

- Make sure that in the **Properties** window for the new cell, you select the **In Use** setting for the **Cell Usage** property.

12. Click **Add** to add other loads to the association.

13. On the **Find** dialog box, in the **Results** data window, select the loads that you require.

💡 **Tips**

- For further details about selecting items, the *Find dialog box*, page 580.

- The software finds only those items that have the **Is Load** property in the **Properties** window set to **True** or if this item is defined as a **Consumer** in the **Common Properties** dialog box. Also, the **Find** dialog box displays only those load items that aren't connected to a bus.

14. Under **Electrical consumption using coincidence factors**, select one of the following calculation modes:

- **For direct loads only** - to calculate the total consumed power only for those loads that are connected directly to the current bus.

- **For all feeders** - to calculate the total consumed power for all the loads fed by the current bus as well as the loads that are fed by the buses that are connected to the current bus (as specified in Options Manager). When you select this option, the **Circuits** tab becomes available.

💡 **Tip**

- You can set a preference that instructs the software to account for transformer power losses. For details, see *Customize All Feeder Load Summary Report Preferences*, page 116.

15. To associate loads with a bus that have been associated only with a PDB, click the **Unassigned Loads** tab.

16. Under **Unassigned loads**, select the item tag type and the **Select** check box for each load that you want to associate.

17. Click **Assign to Bus**.

   💡 **Tip**

   - The software moves the selected load to the **Loads** tab.

18. Click **Apply**.

19. Click the **Bus** tab and define the bus rated data.

   💡 **Tip**

   - If, after calculating the rated power for all feeders, you move to the **Bus** tab and change the bus data, make sure that you click **Apply** before you switch back to the **Loads** tab.

20. Click the **Loads** tab and set the load phases.

21. Continue adding loads as you require and then click **OK** when done.

   📝 **Notes**

   - The software associates only those electrical items that you have defined as consumers. To define an electrical item as a consumer, you have to set the **Is Load** property in the **Properties** window for that item to **True** or define this electrical item as a **Consumer** in the **Common Properties** dialog box of that item.

   - You can assign a load to a particular phase only if you defined the properties of the current bus as having an AC supply and you set a value for the number of phases.

   - The software displays the calculated consumption values according to the loads associated with each bus for the selected phase. If you do not select any bus from the **Bus** list, the software calculates the consumption for the entire power distribution board.

   - The software can account for converting equipment power losses if you have set your report preferences accordingly. For details, see *Customize All Feeder Load Summary Report Preferences*, page 116.

   - If you associate a capacitor, the software subtracts the consumed kVAr value of that capacitor from the total connected consumed kVAr value. Therefore, the consumed electrical power (kVAr) may display a negative value. This means that there is a lagging power factor.

   - If a particular load associated with a selected PDB or bus contains incomplete data, the software displays an appropriate message in the **Note** box.

   - To cancel an association of an item, under **Associated loads**, select a row and click **Remove**.

   - You cannot dissociate a load if it is connected to a circuit.

Associate a Single Load with a PDB

1. Click **Actions** > **Batch Load Association**.

2. On the **Batch Load Association** dialog box, under **Power distribution board**, select a power distribution board.

3. From the **Bus** list, select a bus with which you want to associate the loads.

   💡 **Tip**

   - You do not have to select a bus if you do not want to associate any loads with a specific bus under the selected PDB. In this case, the software calculates the load consumption for the entire power distribution board rather than for a specific bus.

4. Click the **Loads** tab.

5. Click **Add** to open the **Find** dialog box.

6. From the **Equipment type** list, select the desired equipment type.

7. Click **Find Now**.

   💡 **Tip**

   - The software finds only those load items that have the **Is Load** property in the **Properties** window set to **True** or if this load is defined as a **Consumer** in the **Common Properties** dialog box. Also, the **Find** dialog box displays only those load items that are not connected to a bus.

8. From the data window, highlight the desired load item tag.

9. Click **OK** to associate the load with the selected PDB.

10. Under **Create circuits and cells**, select the **Create circuits** check box to automatically create a circuit for each associated load that has not been assigned to a circuit. Note that this option does not affect the loads that have already been assigned to circuits.

    💡 **Tip**

    - If you do not select the check boxes under **Create circuits and cells**, the software associates the selected loads with a power distribution board or a bus. You can create circuits at a later time.

11. Select the **Create cell** check box to automatically create a cell where the software will create the new feeder circuit for the associated load.

    💡 **Tip**

    - Make sure that in the **Properties** window for the new cell, you select the **In Use** setting for the **Cell Usage** property.

12. Under **Associated loads**, scroll to the right and define the phase association of the current load by selecting the appropriate **Phase** check boxes.

13. Click **Apply** and then click the **Bus** tab.

14. Define the bus rated power as needed.

15. To associate loads with a bus that have been associated only with a PDB, click **Unassigned Loads**.

16. Under **Unassigned loads**, select the item tag type and the **Select** check box for each load that you want to associate.

17. Click **Assign to Bus**.

18. Click **Apply** and then select another PDB or another bus.

19. Continue adding loads as you require and then click **OK** when done.

Notes

- You can assign a load to a particular phase only if you defined the properties of the current bus as having an AC supply and you set a value for the number of phases.

- To cancel an association of an item, under **Associated loads**, select a row and click **Remove**. You cannot dissociate a load if it is connected to a circuit.

- The software displays the calculated consumption according to the loads associated with each bus. If you do not select any bus from the **Bus** list, the software calculates the consumption for the entire power distribution board.

- If you associate a capacitor, the software subtracts the rated kVAr value of that capacitor from the total connected rated kVAr value. Therefore, the rated electrical power (kVAr) may display a negative value. This means that there is a lagging power factor.

- If a particular load associated with a selected PDB or bus contains incomplete data, the software displays an appropriate message in the **Note** box.

Related Topics

Display Bus Load Data

1. Click **Actions** > **Batch Load Association**.

2. On the **Batch Load Association** dialog box, under **Power distribution board**, select a power distribution board.

3. From the **Bus** list, select a specific bus.

4. Click the **Loads** tab.

5. Under **Bus phases**, select **All phases** to see all the calculated electrical data for all the associated loads.

   📝 **Notes**

   *   The software displays the calculated electrical consumption for the selected bus.

   *   To view the calculated electrical consumption for a particular phase, select the check boxes as you require in the **Bus phases** group box.

   Related Topics
   *   *Associate a Single Load with a PDB*, page 221
   *   *Associate Loads with a PDB in Batch Mode*, page 219
   *   *Automatic Creation of Feeder Circuits*, page 218
   *   *Create a Bus*, page 154
   *   *Create a Power Distribution Board (PDB)*, page 152
   *   *Managing Power Distribution Boards Common Tasks*, page 151

Associate Bus Riser Circuits with Coupler Circuits

1. In the **Electrical Engineer**, find the bus riser and coupler circuits that you want to associate.

2. Select the bus riser circuit and drag it to the coupler circuit.

   💡 **Tips**

   *   Also, you can use a cable to connect a bus riser with a coupler. To do this, in the **Electrical Engineer**, drag the desired cable to a coupler. Then, drag a sectionizer to that cable. Note that you can use only a single cable and not a series of cables for this purpose.

   *   To facilitate the drag-and-drop operation and to eliminate the need to scroll up or down in the **Electrical Engineer**, open another **Electrical Engineer** window and navigate to the desired location in the tree view before you drag an item.

   Related Topic
   *   *Associating Electrical Equipment Common Tasks*, page 215

Associate a Control Station with an Item

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels** > **Control Stations**.

2. Right-click a control station and click **Common Properties** to open the **Control Station Common Properties** dialog box.

3. Beside **Associated load**, click **Find** to open the **Find** dialog box.

4. From the **Equipment type** list, select the desired equipment type.

5. Click **Find Now**.

6. From the data window, highlight an item tag.

7. Click **OK** to associate the item tag with the current control station.

📝 Notes

- You can associate a control station with a load, converting equipment item, disconnect electrical equipment item, a generator, and a battery bank.

- You can also associate a control station with an electrical item using the **Apply Options** feature for the electrical item with which you want to associate a control station. If you apply a control station to an item that is connected to a circuit, the software associates the control station cables (instrumentation and control) with the circuit automatically when you select the appropriate Options Manager setting. If you do not use this setting, you need to make the association manually. For details, see *Associate Control Station Cables with Equipment Circuits*, page 227.

- You can associate only one control station with an electrical item.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Control Station to an Item*, page 144
- *Associating Electrical Equipment Common Tasks*, page 215
- *Create a Control Station*, page 176
- *Creating Reference Data Common Tasks*, page 125

Associate a Control Station with Cables

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels** > **Control Stations**.

2. Right-click a control station and click **Common Properties** to open the **Control Station Common Properties** dialog box.

3. Click **Add**.

4. On the **Find** dialog box, click **Find Now** to display the available cables.

5. Select a cable to associate with the current control station and click **Apply**.

6. Repeat the previous step for each additional cable that you want to associate with the control station.

   ✎ **Notes**

   - The **Find** dialog box displays only control and grounding cables that have at least one unassociated side.

   - You can associate one control station with more than one cable.

   - You cannot associate a control station with a power cable.

   Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Apply a Typical Control Station to an Item*, page 144
   - *Associating Electrical Equipment Common Tasks*, page 215
   - *Create a Control Station*, page 176
   - *Creating Reference Data Common Tasks*, page 125

Associate Control Station Cables with Equipment Circuits

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Panels** > **Control Stations**.

2. In the lower pane, select one or more control stations.

3. Click **Actions** > **Cables** > **Associate Cables with Equipment Circuits**.

   Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Apply a Typical Control Station to an Item*, page 144
   - *Associate a Control Station with an Item*, page 226
   - *Associating Electrical Equipment Common Tasks*, page 215
   - *Create a Control Station*, page 176

Associate Instrument Cables with Equipment Circuits

1. In the **Electrical Index**, expand the folder hierarchy **Electrical Equipment** > **Instruments**.

2. In the lower pane, select one or more control stations.

3. Click **Actions** > **Cables** > **Associate Cables with Equipment Circuits**.

   Related Topics
   - *Apply a Profile to an Item*, page 147
   - *Associating Electrical Equipment Common Tasks*, page 215

Associate a Power Cable with Electrical Equipment

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Under the folder **Power Cables**, select the desired cable and drag it under a feeder circuit 🗐 in the **Electrical Engineer**.

3. In the **Electrical Engineer**, right-click the cable, and on the shortcut menu, click **Common Properties**.

4. Click the **Design Data** tab and note that the circuit path appears under **From**, in the **Tag** box.

5. Close the dialog box, and in the **Electrical Index**, select the item of electrical equipment with which you want to associate the cable.

    💡 **Tip**

    *   You can select any load that does not already have an association in the **Electrical Engineer**. For the rules that govern the conditions under which you can drag items onto other items, see *Drag-and-Drop Rules from the Electrical Index to the Electrical Engineer*, page 83.

6. Drag the item under the cable in the **Electrical Engineer**.

7. Open the dialog box for the cable again and on the **Design Data** tab, note that the item tag for the equipment appears under **To**, in the **Tag** box.

📝 **Notes**

*   If an item already appears in the **Electrical Engineer**, you can drag a cable onto the item where this is allowed.

*   You can associate a power cable with a motor by defining a motor profile with that cable and then applying the profile to the desired motor.

*   Changing the association of a terminated cable by dragging it to another equipment item in the **Electrical Engineer**, disconnects this cable from the terminal strip of that equipment. The software disconnects the cable without displaying a warning message.

Related Topics

Associate Additional Power Cables with Electrical Equipment

1. In the **Electrical Engineer**, expand the hierarchy to the load to which you want to add one or more cables.

2. Right-click the cable associated with the load, and on the shortcut menu, click **Add Cable**.

3. On the **Find** dialog box, click **Find Now** to display the available power cables.

4. From the list of power cables, while holding **Ctrl**, highlight each cable that you want to add.

5. Click **OK**.

6. In the **Electrical Engineer**, right-click a cable that you added.

7. On the shortcut menu, click **Common Properties**.

8. Click the **Design Data** tab and note that the associated load and feeder path appear in the **From** and **To** sections, in the **Tag** boxes.

📝 Notes

- Cables that you associate in this way do not appear in the parallel cable list in the **Cable Common Properties** dialog box of other cables connected to the same load.

- If you dissociate a cable from the feeder, the software automatically dissociates that cable from the load and dissociates all the other parallel cables from the feeder; however, those cables remain associated with the load.

- Changing the association of a terminated cable by dragging it to another equipment item in the **Electrical Engineer**, disconnects this cable from the terminal strip of that equipment. The software disconnects the cable without displaying a warning message.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Create a Profile*, page 61
- *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81
- *Dissociate an Item from its Feeding Item*, page 90

Associate a Non-Power Cable with Electrical Equipment

1. In the **Electrical Index**, expand the folder hierarchy **Wiring Equipment** > **Cables**.

2. Expand one of the folders **Control Cables**, **Grounding Cables**, or **Instrumentation Cables** and select a cable.

3. Do one of the following:

   - Click **Edit** > **Common Properties**.

   - Right-click the cable and on the shortcut menu, click **Common Properties**.

   - Press **Ctrl** + **F2**.

4. On the **Cable Common Properties** dialog box, click the **Design Data** tab.

5. Click the ellipsis button [...] beside the **From** property to define one side of the cable.

6. On the **Find** dialog box, from the **Equipment type** list, select the desired equipment type, and click **Find Now**.

7. From the data window, highlight an item to select it, and click **OK**.

8. Click the ellipsis button [...] beside the **To** property to define the other side of the cable.

Note

- Changing the association of a terminated cable with another equipment item on the **Cable Common Properties** dialog box, disconnects this cable from the terminal strip of that equipment. The software disconnects the cable without displaying a warning message.

Related Topics

Create a Daisy-Chain Connection Pattern

It is possible to connect a number of electrical items in a daisy-chain pattern. You connect electrical items in a daisy-chain pattern by dragging an item to another item of the same type from the **Electrical Index** to the **Electrical Engineer** or within the **Electrical Engineer**. You can connect the following items in a daisy-chain pattern:

- **Loads** - drag a load associated with a cable to another load in the **Electrical Engineer**.

- **Heat traces** - drag a heat trace to another heat trace in the **Electrical Engineer**. You can also drag a heat tracer to a junction box but not to a circuit in a junction box. Heat traces and miscellaneous electrical equipment items do not require associated cables to male daisy-chain connections.

- **Miscellaneous electrical equipment items** - drag a socket outlet, lighting fixture, welding outlet, and so forth to another item belonging to the Miscellaneous Electrical Equipment type. Also, you can drag an item belonging to the Miscellaneous Electrical Equipment type to a junction box but not to a junction box circuit.

- **Junction boxes** - drag a junction box (but not its circuit) to a heat trace or to an item belonging to the Miscellaneous Electrical Equipment type. There is no need for a cable to be associated with a junction box.

Notes

- On the **Batch Load Association** dialog box, the software displays items connected in a daisy-chain pattern as if they are all connected in a parallel pattern. The software handles both parallel and daisy-chain connections in the same way.

- On the **Common Properties** dialog box, items connected in a daisy-chain pattern appear as if they are fed from the same feeder circuit.

- You cannot drag loads that are connected in a daisy-chain pattern to the same circuit to create a parallel connection. However, you can drag them to other circuits.

Related Topics

Balancing Loads Common Tasks

Load balancing is an operation that you can perform at any stage of your project; however, it is recommended that you perform load balancing before you create feeder cables between loads and their circuits, and before you apply profiles to the loads. The reason for this is that when you move loads between buses, associated power cables remain attached to the previous feeder circuits. The following tasks are used frequently when you balance loads between buses.

Validate Bus Loads

This procedure shows how to define current settings and requested spare capacity for bus load lines and to perform validations of bus loads. For more information, see *Validate Bus Loads*, page 233.

Balance Bus Loads

This procedure shows how to move loads between different buses in your project. For more information, see *Balance Bus Loads*, page 233.

Related Topics
- *Balance Bus Loads*, page 233
- *Validate Bus Loads*, page 233

Validate Bus Loads

1. Associate loads to buses individually or in batch mode. For more information, refer to Related Topics.

2. Click **Actions** > **Batch Load Association**.

3. On the **Batch Load Association** dialog box, under **Power distribution board**, select a power distribution board.

4. From the **Bus** list, select a specific bus.

5. Click the **Bus** tab.

6. Type values in the **Line current** and **Overload percentage** boxes.

7. Enter or modify other values as desired.

8. Close the **Batch Load Association** dialog box.

9. In the **Electrical Index**, select one or more PDBs or buses.

10. Click **Actions** > **Total Bus Load Validation**.

   💡 **Tip**

   - The software calculates the load data for the selected buses and displays the results on the **Total Bus Load Validation** dialog box.

11. If desired, click **Excel Report** to view the results in an Excel file.

📝 **Notes**

- The software uses the total bus amperes as the basis for calculating the total bus load, including any loads connected downstream. The input for the calculation comes from the consumed active and reactive loads.

- The software does not perform bus load validation for loads that have incomplete data, that is, where any of the following properties has a value of zero:

- The consumed active load.

- The consumed reactive load.

- The coincidence factor (X, Y, or Z) that applies for the load.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Electrical Equipment with Feeders*, page 216
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Bus Tab (Batch Load Association Dialog Box)*, page 667
- *Change an Existing Electrical Association*, page 85
- *Create Electrical Associations Using the Drag-and-Drop Operation*, page 81

Balance Bus Loads

1. Click **Actions** > **Batch Load Association**.

2. On the **Batch Load Association** dialog box, under **Power distribution board**, select a power distribution board.

3. From the **Bus** list, select a specific bus.

4. Click the **Loads** tab.

5. Select the load that you want to move.

6. Click **Move to Cache**.

   💡 **Tip**

   - The software moves the load from **Associated loads** list to the **Cached Loads** tab.

7. Repeat the two previous steps for each load that you want to dissociate from the currently selected bus.

8. Click the **Cached Loads** tab.

9. Click **Apply**.

10. From the **Power distribution board** and **Bus** lists, selected the desired PDB and bus to which you want to move the cached loads.

   💡 **Tips**

   - On performing this step, the software prompts you to save the changes. If you click **Yes**, the software dissociates the cached loads from the bus to which they were assigned, and you can now reassign them.

   - Also, the software displays a message allowing you to move the selected bus together with its feeding circuit and the circuit internal items. However, if the load that you want to move is connected to the bus through converting equipment, the feeding circuit cannot be moved with the load.

11. Do one of the following:

   - For each load that you want to reassign, select the check box in the **Item Tag** column.

   - To reassign all of the loads in the list, select the **Select all** check box.

12. Click **Assign to Bus**.

13. If you have not assigned all the items to new buses, repeat the three previous steps for each load that you want to reassign.

   📝 **Note**

   - The **Cached Loads** tab behaves as a temporary data store only; the software removes all loads from the list when you close the dialog box.

Related Topics
   - *Change an Existing Electrical Association*, page 85
   - *Validate Bus Loads*, page 233

Calculating Bus Load Summaries: An Overview

SmartPlant Electrical enables you to calculate bus load totals on a specific power distribution board, bus, or on all the buses in the plant. This calculation facilitates the sizing of electrical equipment such as generators, distribution transformers and their feeder cables, buses, UPSs and so forth. After completing the calculation, SmartPlant Electrical stores the results in the database. You can calculate the bus load totals for all the existing buses in the plant or just for those PDBs or buses that you select in the **Electrical Index** or the **Electrical Engineer**. Once the software completes the calculation, you can generate a load summary report for the calculated loads based on the calculated and stored properties.

Electrical Load Data

When performing a load summary calculation, the software uses the values of the following load properties:

- Rated active (kW at rated/nominal/100% load conditions)

- Rated reactive (kVAR at rated/nominal/100% load conditions)

- Consumed active (kW at operating point/consumed load conditions)

- Consumed reactive (kVAR at operating point /consumed load conditions)

- Operating mode (Continuous, Intermittent, Standby or Spare)

- Coincidence factors for each and every mode of operation (x,y,z,zz)

Compensated and Uncompensated Electrical Load Values

The software uses two sets of data for load values:

- **Uncompensated values**— rated and consumed electrical load kW and kVAR values without applying coincidence factors.

- **Compensated values**— rated and consumed electrical load kW and kVAR values multiplied by their respective individual coincidence factors.

Example:

Let's assume that an intermittent load that has the following values:

- Rated active power = 100 kW
- Rated reactive power = 10 kVAR
- Consumed active power = 80 kW
- Consumed reactive power = 6 KVAR
- Y = 0.8
- X = 1

The uncompensated values would as follows:

- Uncompensated rated active power = 100 kW
- Uncompensated rated reactive power = 10 kVAR
- Uncompensated consumed active power = 80 kW
- Uncompensated consumed reactive power = 6 kVAR

The compensated active power values would be as follows (uncompensated values multiplied by the coincidence factor):

- Compensated rated active power = 80 kW
- Compensated rated reactive power = 8 kVAR
- Compensated consumed active power = 64 kW
- Compensated consumed reactive power = 4.8 kVAR

Notes

1. Bus and circuit totals depend on the settings of your preferences. Therefore, run the bus load calculation to update the total values in the bus, circuit, and load properties.

2. Loads are accounted for whether or not they are connected through circuits.

3. Circuit connection mode (**Connected** or **Disconnected**) affects the calculation results. Disconnecting a circuit excludes the connected load from the calculation.

4. The circuit properties store the total value of the load that is connected to it as follows:

   a. The properties of circuits that feed direct loads store the load data.

   b. The properties of circuits that feed downstream buses whether directly or through converting equipment store the downstream bus load total values. Practically, these values will be the same as for the buses that are fed by these circuits.

   c. The properties of coupler circuits store the total values of the bus is connected to the corresponding riser provided that your preferences have been set do so. If your preferences are set to exclude the couplers, the calculation results for the coupler circuits will be null.

   d. The properties of riser circuits store the total values of the bus that is connected to the feeding coupler. If your preferences are set to exclude risers, the calculation results for riser circuits will be null.

   e. If your preferences are set to account for converting equipment losses, the calculation results will include the kVAR and kW values.

For further information, see the following topics:

Connectivity Across Buses, page 236

Calculating Bus Loading, page 239

Correcting Bus Power Factors Using Capacitor Banks, page 242

Bus Load Calculation Preferences, page 245

Handling Special Cases of Bus Load Calculations, page 247

Calculating Electrical Consumption During Batch Load Assignment, page 248

Flow of Activities for Bus Load Calculations, page 249

Bus, Circuit, and Converting Equipment Properties, page 250

Related Topics
- *All Feeder Load Summary (Enhanced) Report*, page 404
- *Calculating Bus Load Summaries Common Tasks*, page 251

Connectivity Across Buses

Calculating bus loading basically means summing up all the loads that are connected a specific bus.
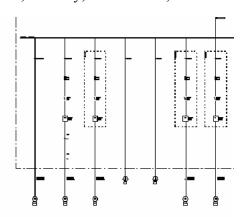
Circuit Mode

The calculation of bus load summary depends on the mode of the circuits. The circuit mode property can be **Connected** or **Disconnected** and it controls whether the loads are connected or disconnected to the bus. You set the circuit mode on the **Circuit Common Properties** dialog box or in the **Properties** window. Setting the circuit mode to **Connected** means that power flows through the circuit and it acts as a connecting line between the input and output of the bus. Setting the circuit mode to **Disconnected** interrupts the circuit connectivity and it means the power is not flowing therefore, the downstream connected loads or buses are not accounted for in the calculation.

The following situations can arise when identifying the buses that are connected to a particular bus:
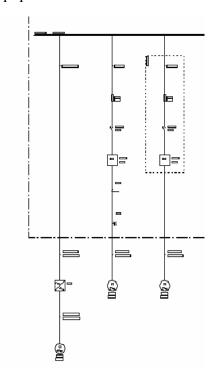
Loads Connected Directly to a Bus

As shown in the diagram below, the loads are connected directly to the bus, which means that they are connected to the bus via a circuit or directly without a cable. A load can be connected via a junction box or any other connecting item such as a panel, bus way, and so forth, all of which are transparent to the bus load association.
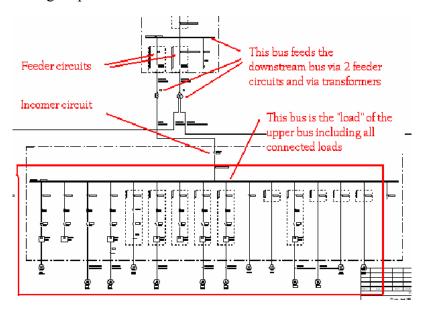
Loads Connected Indirectly to a Bus (Via Converting Equipment)

In this case, the loads are connected to a bus indirectly. That is, the loads are connected to transformer , a variable frequency drive, or any other converting equipment item within the circuit internals or external to the circuit.

Buses Connected Via Feeder-Incomer Circuits
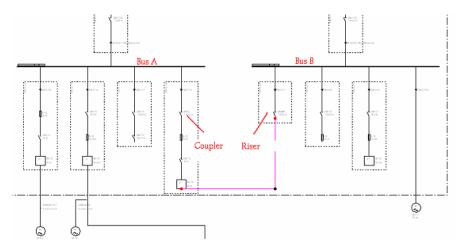
The diagram below shows a configuration of buses that feed downstream buses through a pair of feeder-incomer circuits

As shown above, the loads that connect the upper bus are the direct loads as well as all the downstream connected buses which in turn also have loads that are connected to them directly. Any calculation of the upper bus also accounts for the downstream connected loads. Note that the software does not impose any limitation on the number of downstream levels of buses for a network. The software makes a recursive calculation.

Buses Connected Via Coupler-Riser Circuits

Buses can also be connected to each other by sets of coupler-riser circuits. Such a connection is usually used for connecting buses of the same voltage level mainly for redundancy and backup purposes when one bus can take over the loads of the coupled bus in case of loss power source in one of the adjacent buses.

Bus A and bus B are coupled (connected) by the coupler and riser circuits which in principle serve as mutual backups for power supply.

Related Topics

- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Calculating Bus Loading

Calculating bus loading basically means summing up all the loads that are connected a particular bus. When calculating the total loads connected to a bus, the software calculates the bus totals, stores the individual subtotals in the database as calculated properties and then makes these calculated properties accessible for reporting. During this stage, you can sum them up any way you see appropriate.

In general, the total load of a bus is a variation of the sum of its active and reactive vectors. SmartPlant Electrical calculates these totals for each load by adding up the direct and downstream rolled up loads. Then, SmartPlant Electrical stores these results in the database.

Bus Values Used in Calculations

Bus Rated Power Values

- Total uncompensated continuous active power
- Total uncompensated continuous reactive power
- Total uncompensated intermittent active power
- Total uncompensated intermittent reactive power
- Total uncompensated standby active power
- Total uncompensated standby reactive power
- Total uncompensated spare active power
- Total uncompensated spare reactive power
- Total compensated continuous active power
- Total compensated continuous reactive power
- Total compensated intermittent active power
- Total compensated intermittent reactive power
- Total compensated standby active power
- Total compensated standby reactive power
- Total compensated spare active power
- Total compensated spare reactive power

Bus Consumed Power Values

- Total uncompensated continuous active power
- Total uncompensated continuous reactive power
- Total uncompensated intermittent active power
- Total uncompensated intermittent reactive power
- Total uncompensated standby active power
- Total uncompensated standby reactive power
- Total uncompensated spare active power
- Total uncompensated spare reactive power
- Total compensated continuous active power
- Total compensated continuous reactive power
- Total compensated intermittent active power
- Total compensated intermittent reactive power
- Total compensated standby active power
- Total compensated standby reactive power
- Total compensated spare active power
- Total compensated spare reactive power

Circuit Load Calculation

SmartPlant Electrical stores downstream load totals in the respective circuit items. The following rules apply to the data that a circuit stores:

- **Feeder circuits**— hold the values of the total downstream connected load (if there is more than one load connected to a circuit, the circuit stores the total values for all the connected loads).

- **Incomer circuits**— store the total load of the bus that the incomer circuit is feeding. Note that the incomer circuit values and total bus values should be the same.

- **Coupler circuit**— stores the values of the load of the coupled bus.

- **Riser circuits**— store the values of the bus that has the coupler, opposite the riser circuit. That is, each coupler-riser paired circuit stores the values of its counterpart.

Calculating the Total Bus Loading

SmartPlant Electrical calculates the following grand totals of bus loading for each set of the bus rated power values and bus consumed power values:

Rated Power

- Total running bus active load
- Total running bus reactive load
- Total running bus apparent load
- Running bus power factor
- Total peak bus active load
- Total peak bus reactive load
- Total peak bus apparent load
- Peak bus power factor

Consumed Power

- Total running bus active load
- Total running bus reactive load
- Total running bus apparent load
- Running bus power factor
- Total peak bus active load
- Total peak bus reactive load
- Total peak bus apparent load
- Peak bus power factor

The total **running** load is defined as the sigma of the **continuous** and **intermittent** loads, compensated or uncompensated, respectively.

The total **peak** load is defined as the sigma of the **continuous**, **intermittent**, **standby**, and **spare** loads, compensated or uncompensated respectively.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234
- *Connectivity Across Buses*, page 236

Correcting Bus Power Factors Using Capacitor Banks

When running batch load assignment and calculating the electrical consumption for a bus, the software allows you to calculate the power factor of a capacitor bank that needs be associated with the selected bus so that you can correct the power factor of that bus. Creating a capacitor bank that has the power factor you just calculated and associating this capacitor bank with the current bus corrects the bus power factor when calculating the electrical consumption of the bus.

The formula that SmartPlant electrical uses to calculate the required correction factor is as follows:

$$Q_{cb} = Q - P \cdot tg\varphi_d$$

Where Qcb = Reactive Power of the Capacitor Banks to be connected to the bus in order to have the desired cosϕ.

1. On the **Bus** tab of the **Batch Load Association** dialog box, type a value under **Required power factor**. This value determines the required power factor for the selected bus.

2. Click the **Loads** tab.

3. Under **Electrical consumption using coincidence factors**, select the **For all feeders** option to activate the calculation the electrical consumption and the required corrective reactive power for the selected bus.

    💡 **Tip**

    -   If you choose to close the **Batch Load Association** dialog box at this time, you can also run this calculation by clicking **Actions** > **Calculate Bus Loads** > **Selected PDBs or Buses**.

4. On the **Batch Load Association** dialog box, click the **Loads** tab.

5. Under **Required capacitor correction [kVAR]**, copy the calculated value. You can copy either total running load or the total peak load value as you require.

6. Click **OK** on the **Batch Load Association** dialog box.

7. In the **Electrical Index**, create a new capacitor bank. For details, see *Create Static Electrical Equipment*, page 132.

8. On the **Common Properties** dialog box for the new capacitor bank, click the **Load Data** tab.

9. Under **Rated power**, paste in the **Required capacitor correction [kVAR]** value that you copied from the **Batch Load Assignment** dialog box.

10. Click the **Feeder Data** tab and associate the current capacitor bank with the bus for which you ran the electrical consumption calculation.

11. Click **OK** to accept the values you entered and close the **Common Properties** dialog box.

✎ **Note**

- You can now open the **Batch Load Assignment** dialog box and recalculate the bus load consumption for that bus. Under **Power factor@Run** and **Power factor@Peak,** You can see the newly calculated power factor values for the current bus associated with the new capacitor bank while the value under **Required capacitor correction** value is zero or close to zero.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Bus Load Calculation Preferences

SmartPlant Electrical calculates bus load summaries according to the preferences that you set on the **Reports** tab of the **Preferences** dialog box. The options are as follows:

- **Use each load coincidence factor**— When calculating the compensated load values, the software multiplies the raw uncompensated load kW and kVAR values by the load coincidence factors.

- **Use Bus PDB coincidence factor**— When calculating the compensated load values, the software multiplies the raw uncompensated load kW and kVAR values by the coincidence factors as defined in the properties of the buses that these loads are connected to. Make sure that you specify these coincidence factors for each bus.

- **Do not include coupled buses**— When calculating bus totals, the software ignores adjacent couples buses (regardless of whether they are connected via couplers or risers). The software does not account for these adjacent buses and treats them as if they are disconnected.

- **Include coupled buses connected by couplers only**— When calculating bus totals, the software includes adjacent bus loads connected via couplers and excludes the buses connected via risers.

- **Include coupled buses connected by couplers and risers**— When calculating bus totals, the software includes adjacent bus loads connected via couplers and risers (full redundancy).

- **Account for converting equipment power losses** - When running bus load calculations, allows you to instruct the software to calculate the total electrical consumption taking into account converting equipment power losses. The software adds these power losses to the upstream feeding bus and accounts for them as continuous loads. The same losses are also accounted for in the All Feeder Load Summary and PDB Load Summary reports.

Disconnected Circuits

If the mode of a circuit is set to **Disconnected**, SmartPlant Electrical ignores this circuit in the calculation regardless of the preference settings.

Related Topics

Handling Special Cases of Bus Load Calculations

You should pay special attention to the way SmartPlant Electrical treats and accounts for certain item types when calculating bus loading.

- **Three-winding transformers**— A bus feeding a two-winding transformer accounts for both the secondary connected loads and buses.

- **Transfer switch (a special PDB with two incomers and one feeder)**— Make sure that you connect the incomer circuit that you require.

- **Dual power source equipment**— The software treats equipment with two power sources as one load for each of the power paths.

- **Bus Drill Down Enabled property** — If, in the **Properties** window, you set this property to **False**, the software does not roll up any of its connected direct or indirect loads. Instead, the software accounts for the rated and consumed power values that you have entered manually in the **Properties** window of each respective bus.

Related Topics

Calculating Electrical Consumption During Batch Load Assignment

When running batch load association and calculating electrical consumption for all the feeders, SmartPlant Electrical uses the same algorithm as the bus loading calculation. Note that all the values for the electrical consumption in batch load association are consumed and compensated for their coincidence factors whether for individual loads or the current bus as set in the preferences.

Also, the software calculates line current and power factors at run and peak conditions using consumed values.

SmartPlant Electrical calculates electrical consumption for selected a selected bus, all the rolled up buses, and the loads associated with that bus.

The software makes all the calculations based on the phase filter in the batch load association.

As with bus load calculations, SmartPlant Electrical stores the calculated electrical consumption results in the database.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Flow of Activities for Bus Load Calculations

The following is a suggested flow of activities for bus load calculations.

1. In the **Electrical Index**, select the buses or PDBs for which you want to calculate a bus load summary. You can select one bus or multiple buses as the starting point of the calculation and determine which of the coupled buses (connected coupled buses) you want to account for in the calculation.

2. Execute the calculation command. For details, see the following:

   - *Calculate Bus Loads for Selected Buses or PDBs*, page 255

   - *Calculate Bus Loads for All the Plant Buses*, page 255

3. Generate a calculation report for the buses you selected. For details, see *Generate an Enhanced All Feeder Load Summary Report*, page 256.

Note

- To refresh the calculation results, you must recalculate the bus load summary for the required buses and then regenerate the Enhanced All Feeder Load Summary report.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Bus, Circuit, and Converting Equipment Properties

After calculating bus loading, SmartPlant Electrical uses the following bus, circuit, and converting equipment properties to store the calculation results:

Bus Properties

Calculated Line Currents (Using Coincidence Factor Compensated Load Data

- LineCurrent_Normal_Consumed
- LineCurrent_Peak_Consumed
- LineCurrent_Normal_Rated
- LineCurrent_Peak_Rated

Calculated Total Rolled Up Bus Loading @ Using Consumed Load Data Compensated by Coincidence Factors

- ComplRolledupRunActiveLoad_Consumed
- CompRolledupRunReactiveLoad_Consumed
- CompRolledupRunApparentLoad_Consumed
- CompRolledupRunPowerFactor_Consumed
- CompRolledupPeakActiveLoad_Consumed
- CompRolledupPeakReactiveLoad_Consumed
- CompRolledupPeakApparentLoad_Consumed
- CompRolledupPeakPowerFactor_Consumed

Calculated Total Rolled Up Bus Loading @ Using Rated Load Data Compensated by Coincidence Factors

- CompRolledupRunActiveLoad_Rated
- CompRolledupRunReactiveLoad_Rated
- CompRolledupRunApparentLoad_Rated
- CompRolledupRunPowerFactor_Rated
- CompRolledupPeakActiveLoad_Rated
- CompRolledupPeakReactiveLoad_Rated
- CompRolledupPeakApparentLoad_Rated
- CompRolledupPeakPowerFactor_Rated

Calculated Total Rolled Up Bus Loading @ Using Consumed Load Data Without Compensation of Coincidence Factors

- UncompRolledupRunActiveLoad_Consumed
- UncompRolledupRunReactiveLoad_Consumed
- UncompRolledupRunApparentLoad_Consumed
- UncompRolledupRunPowerFactor_Consumed
- UncompRolledupPeakActiveLoad_Consumed
- UncompRolledupPeakReactiveLoad_Consumed
- UncompRolledupPeakApparentLoad_Consumed
- UncompRolledupPeakPowerFactor_Consumed

Calculated Total Rolled Up Bus Loading @ Using Rated Load Data Without Compensation of Coincidence Factors

- UncompRolledupRunActiveLoad_Rated
- UncompRolledupRunReactiveLoad_Rated
- UncompRolledupRunApparentLoad_Rated
- UncompRolledupRunPowerFactor_Rated
- UncompRolledupPeakActiveLoad_Rated
- UncompRolledupPeakReactiveLoad_Rated
- UncompRolledupPeakApparentLoad_Rated
- UncompRolledupPeakPowerFactor_Rated

Calculated Total Rolled Up Bus Loading per Operating Modes @ Using Consumed Load Data with Compensation of Coincidence Factors

- CompRolledupContinuousActiveLoad_Consumed
- CompRolledupContinuousReactiveLoad_Consumed
- CompdRolledupIntermittentActiveLoad_Consumed
- CompRolledupIntermittentReactiveLoad_Consumed
- CompRolledupStandbyActiveLoad_Consumed
- CompRolledupStandbyReactiveLoad_Consumed
- CompRolledupSpareActiveLoad_Consumed
- CompRolledupSpareReactiveLoad_Consumed

Calculated Total Rolled Up Bus Loading per Operating Modes @ Using Rated Load Data with Compensation of Coincidence Factors

- CompRolledupContinuousActiveLoad_Rated
- CompRolledupContinuousReactiveLoad_rated
- CompdRolledupIntermittentActiveLoad_Rated
- CompRolledupIntermittentReactiveLoad_Rated
- CompRolledupStandbyActiveLoad_Rated
- CompRolledupStandbyReactiveLoad_Rated
- CompRolledupSpareActiveLoad_Rated
- CompRolledupSpareReactiveLoad_Rated

Calculated Total Rolled Up Bus Loading per Operating Modes @ Using Consumed Load Data Without Compensation of Coincidence Factors

- UncompRolledupContinuousActiveLoad_Consumed
- UncompRolledupContinuousReactiveLoad_Consumed
- UncompRolledupIntermittentActiveLoad_Consumed
- UncompRolledupIntermittentReactiveLoad_Consumed
- UncompRolledupStandbyActiveLoad_Consumed
- UncompRolledupStandbyReactiveLoad_Consumed
- UncompRolledupSpareActiveLoad_Consumed
- UncompRolledupSpareReactiveLoad_Consumed

Calculated Total Rolled Up Bus Loading per Operating Modes @ Using Rated Loads Data Without Compensation of Coincidence Factors

- UncompRolledupContinuousActiveLoad_Rated
- UncompRolledupContinuousReactiveLoad_Rated
- UncompRolledupIntermittentActiveLoad_Rated
- UncompRolledupIntermittentReactiveLoad_Rated
- UncompRolledupStandbyActiveLoad_Rated
- UncompRolledupStandbyReactiveLoad_Rated
- UncompRolledupSpareActiveLoad_Rated
- UncompRolledupSpareReactiveLoad_Rated

Circuit Properties

Calculated Total Rolled Up Circuit Loading per Operating Modes @ Using Consumed Load Data Without Compensation of Coincidence Factors

- UncompRolledupContinuousActiveLoad_Consumed
- UncompRolledupContinuousReactiveLoad_Consumed
- UncompRolledupIntermittentActiveLoad_Consumed
- UncompRolledupIntermittentReactiveLoad_Consumed
- UncompRolledupStandbyActiveLoad_Consumed
- UncompRolledupStandbyReactiveLoad_Consumed
- UncompRolledupSpareActiveLoad_Consumed
- UncompRolledupSpareReactiveLoad_Consumed

Calculated Total Rolled Up Circuit Loading per Operating Modes @ Using Rated Load Data Without Compensation of Coincidence Factors

- UncompRolledupContinuousActiveLoad_Rated
- UncompRolledupContinuousReactiveLoad_Rated
- UncompRolledupIntermittentActiveLoad_Rated
- UncompRolledupIntermittentReactiveLoad_Rated
- UncompRolledupStandbyActiveLoad_Rated
- UncompRolledupStandbyReactiveLoad_Rated
- UncompRolledupSpareActiveLoad_Rated
- UncompRolledupSpareReactiveLoad_Rated

Converting Equipment Properties

Converting Equipment Properties

- ActiveLosses
- ReactiveLosses

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Calculating Bus Load Summaries Common Tasks

To facilitate the sizing of electrical equipment such as generators, distribution transformers and their feeder cables, buses, UPSs and so forth, SmartPlant Electrical enables you to calculate bus load totals and store the results in the database. The software provides for a fast way to calculate the bus load totals for all the existing buses in the plant. Alternatively, you can select the PDBs or buses for which you want to perform the bus load calculation. SmartPlant Electrical stores the calculation results in the database thus allowing you to generate an all feeder load summary report for the calculated loads.

Calculate Bus Loads for Selected Buses or PDBs

This procedure shows how to calculate load totals for selected buses or PDBs. For more information, see *Calculate Bus Loads for Selected Buses or PDBs*, page 255.

Calculate Bus Loads for All the Plant Buses

This procedure shows how to calculate load totals for all the buses in your plant. For more information, see *Calculate Bus Loads for All the Plant Buses*, page 255.

Generate an Enhanced All Feeder Load Summary Report

This procedure shows how to generate an enhanced all feeder load summary report for the selected PDBs or buses. The software generates the report based on the values that it calculated for the PDBs or buses that were saved to the database after the calculation. For more information, see *Generate an Enhanced All Feeder Load Summary Report*, page 256.

Calculate Bus Loads for Selected Buses or PDBs

1. In the **Electrical Index**, select the buses or PDBs for which you want to make the calculation.

2. Click **Actions** > **Calculate Bus Loads** > **Selected PDBs or Buses**.

📝 **Notes**

- The software makes the calculation and stores the results in the database.

- You can now generate an enhanced all feeder load summary for the selected PDBs or buses.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Calculate Bus Loads for All the Plant Buses

- Click **Actions** > **Calculate Bus Loads** > **All Plant Buses**.

Notes

- The software makes the calculation and stores the results in the database.

- You can now generate an enhanced all feeder load summary for the plant PDBs or buses.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Generate an Enhanced All Feeder Load Summary Report

1. In the **Electrical Index**, do one of the following:

   - Select all power distribution boards in your plant.

   - Select the buses or PDBs for which you ran the bus load calculation.

2. Click **Reports** > **Plant Reports**.

3. On the **Plant Reports** dialog box, select **All Feeder Load Summary Report (Enhanced)** and click **OK**.

4. On the **Report Preferences** dialog box, select the following options:

   - To account for the coincidence factors, select **Compensated values**.

   - To ignore coincidence factors, select **Uncompensated values**.

   - To display rated electrical consumption, select **Rated values**.

   - To display consumed electrical consumption, select **Consumed values**.

5. Click **OK**.

Related Topics
- *Calculating Bus Load Summaries Common Tasks*, page 251
- *Calculating Bus Load Summaries: An Overview*, page 234

Metering Equipment: An Overview

SmartPlant Electrical allows you to create equipment for measuring electrical parameters such as voltages and currents. The software allows you to create voltmeters, ammeters, and multimeters. For measuring large voltages and currents, you can also create measuring transformers for these parameters. You create standard meters and measuring transformers in the Reference Data Explorer as standalone items or as part of reference circuits.

In addition, you can create protection relays with various functions for protecting the circuit.

In the **Electrical Index**, you can locate metering equipment at any level under a PDB. You can display metering equipment in single line diagrams.

Related Topics
- *Using Metering Equipment Common Tasks*, page 257

Using Metering Equipment Common Tasks

The following tasks are used frequently when you use metering equipment in your electrical layout.

Create Measuring Transformers

This procedure explains how to create current and potential transformers. These items are 'in-line' items which you can pace at specific points in a circuit to measure currents and voltages. For more information, see *Create Measuring Transformers*, page 258.

Create Meters

This procedure explains how to create ammeters, voltmeters, or multimeters. You can use these meters to measure the outputs of measuring transformers. For more information, see *Create Meters*, page 259.

Create Relay Functions

You create relay functions in the **Reference Data Explorer**. Each relay function represents a default action in response to a standard setting. After you add relay functions to protection relays in the **Electrical Index**, you can modify the function parameters. For more information, see *Create Relay Functions*, page 260.

Create Protection Relays

This procedure explains how to create protection relays. You can assign one or more functions to a protection relay. For more information, see *Create Protection Relays*, page 261.

Copy Metering Equipment

This procedure explains how to copy metering equipment by dragging default items from the **Reference Data Explorer** to an appropriate target folder in the **Electrical Index**. For more information, see *Copy Metering Equipment*, page 262.

Related Topics
- *Metering Equipment: An Overview*, page 256

Create Measuring Transformers

1. Do one of the following:

    - In the **Reference Data Explorer**, expand the hierarchy **Power Distribution Equipment** > **Disconnect Electrical Equipment** > **Metering Equipment**.

    - In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders, and then select an item according to the following step.

2. Do one of the following:

    - Under the **Electrical Equipment** folder, navigate to a load, an item of converting equipment, or another electrical item, and display the **Metering Equipment** folder for that item.

    - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a **Metering Equipment** folder under a power distribution board, a bus, a cell, or a transfer switch.

    - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a circuit.

3. Right-click the selected item, and then click one of the following:

    - **New Current Transformer**.

    - **New Potential Transformer**.

4. Select the new item that you have created, and in the **Properties** window, type a numeric value for the **Tag Sequence No** property.

    💡 **Tip**

    - The **Tag Sequence No** property determines the positioning of the measuring transformer on the SLD.

5. With the item selected, do one of the following:

    - Click **Edit** > **Common Properties**.

    - Right-click the item and on the shortcut menu, click **Common Properties**.

    - Press **Ctrl + F2**.

6. On the **Metering Equipment Common Properties** dialog box, enter the desired values for the item.

Related Topics

Create Meters

1. Do one of the following:

   - In the **Reference Data Explorer**, expand the hierarchy **Power Distribution Equipment** > **Disconnect Electrical Equipment** > **Metering Equipment**.

   - In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders, and then select an item according to the following step.

2. Do one of the following:

   - Under the **Electrical Equipment** folder, navigate to a load, an item of converting equipment, or another electrical item, and display the **Metering Equipment** folder for that item.

   - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a **Metering Equipment** folder under a power distribution board, a bus, a cell, or a transfer switch.

   - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a circuit.

3. Right-click the selected item, and then click one of the following:

   - **New Ammeter**.

   - **New Voltmeter**.

   - **New Multimeter**.

4. Select the new item that you have created, and in the **Properties** window, type a numeric value for the **Tag Sequence No** property.

   💡 **Tips**

   - The **Tag Sequence No** property determines the positioning of the meter on the SLD.

   - To display a meter on an SLD, you need to drag it to the SLD after generation.

5. With the item selected, do one of the following:

   - Click **Edit** > **Common Properties**.

   - Right-click the item and on the shortcut menu, click **Common Properties**.

   - Press **Ctrl** + **F2**.

6. On the **Metering Equipment Common Properties** dialog box, enter the desired values for the item.

7. For a multimeter, click the **Ranges** tab to specify the meter functions and desired range of values for each function.

Create Relay Functions

1. In the **Reference Data Explorer**, select the **Electrical Equipment** folder and select **Relay Functions**.

2. Right-click the folder, and then click **New Relay Function**.

3. In the **Properties** window, enter values for the following relay function properties:

   - **Relay Action**.

   - **Relay Function Code**.

   - **Relay Function Type**.

   - **Relay Percent Setting**.

   - **Relay Setting**.

   💡 **Tip**

   - The software uses the values that you enter for the relay functions as defaults when you add those relay functions to a protection relay. Any subsequent modifications you make to those values in the protection relay do not change the default relay function values.

Related Topics
- *Copy Metering Equipment*, page 262
- *Metering Equipment: An Overview*, page 256
- *Using Metering Equipment Common Tasks*, page 257

Create Protection Relays

1. Do one of the following:

   - In the **Reference Data Explorer**, expand the hierarchy **Power Distribution Equipment** > **Disconnect Electrical Equipment** > **Metering Equipment**.

   - In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders, and then select an item according to the following step.

2. Do one of the following:

   - Under the **Electrical Equipment** folder, navigate to a load, an item of converting equipment, or another electrical item, and display the **Metering Equipment** folder for that item.

   - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a **Metering Equipment** folder under a power distribution board, a bus, a cell, or a transfer switch.

   - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a circuit.

   - Expand the hierarchy **Power Distribution Equipment** > **Disconnect Electrical Equipment** > **Protection Relays**.

3. Right-click the selected item, and then click **New Protection Relay**.

4. Select the new protection relay that you have created, and in the **Properties** window, type a numeric value for the **Tag Sequence No** property.

   💡 **Tip**

   - The **Tag Sequence No** property determines the positioning of the measuring transformer on the SLD.

   - To display a protection relay on an SLD, you need to drag it to the SLD after generation.

5. With the protection relay selected, do one of the following:

   - Click **Edit** > **Common Properties**.

   - Right-click the protection relay and on the shortcut menu, click **Common Properties**.

   - Press **Ctrl + F2**.

6. On the **Metering Equipment Common Properties** dialog box, enter the desired values for the protection relay.

7. Click the **Functions** tab to specify the relay functions that you want to include.

8. Click **Add** for each new function that you want to add to the relay and from the **Find** dialog box, select the desired relay function.

9. Modify the function properties if desired.

    💡 **Tip**

    *   Modifying the function properties does not affect the default values for that function.

Related Topics
*   *Copy Metering Equipment*, page 262
*   *Metering Equipment: An Overview*, page 256
*   *Using Metering Equipment Common Tasks*, page 257

Copy Metering Equipment

1. In the **Reference Data Explorer**, expand the folder hierarchy **Electrical Equipment** > **Power Distribution Equipment** > **Disconnect Electrical Equipment** > **Metering Equipment**.

2. Select an item in one of the sub-folders.

3. In the **Electrical Index**, expand the folder hierarchy and display the **Metering Equipment** folder for one of the following item types:

    *   Loads

    *   Converting equipment

    *   Converting equipment components

    *   Battery banks

    *   Generators

    *   Off-site power

    *   PDBs

    *   Buses

    *   Cells

    *   Circuits

    *   Disconnect electrical equipment

    *   Transfer switches

4. Drag the desired metering equipment item from the **Reference Data Explorer** to the appropriate target folder.

Related Topics
*   *Create Meters*, page 259
*   *Create Relay Functions*, page 260
*   *Using Metering Equipment Common Tasks*, page 257

Move an Item to Another Plant Group

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Select one or more items of the same item type.

3. Click **Actions** > **Move Items** to open the **Move Items to Plant Group** dialog box.

4. Expand the plant hierarchy and then select the plant group that you require.

5. At the prompt, click **Yes** to confirm the move.

📝 Notes

- You must have appropriate access rights for the target plant group to which you want to move items.

- You can move any item type in any folder that is under the **Electrical Equipment** folder. The software does not move any control stations, circuits, and so forth that are associated with a moved item.

- If you move an item that has an I/O signal that is in the same plant group as that item, the software moves the signal with the item to the target plant group. If the signal is in a plant group that is different from the item's plant group, when you move the item the signal remains in its original plant group.

- If you move a motor with a space heater, the software moves the space heater with the motor. You cannot move space heaters independently.

- If you move a cable, the software also moves its conductors or cable sets. You cannot move these cable components independently. Also, if you move one cable that is part of a parallel cable assembly, the software moves all the cables in the assembly.

- If you move a junction box or a local panel, the software moves any circuits associated with the moved item.

- You can move a PDB with its associated components, such as buses, cells, and circuits; however you cannot move those components independently.

- You can only move disconnect electrical equipment items that are not associated with any circuits.

Related Topics

- *Move Items to Plant Group Dialog Box*, page 580

Working with Drawing Templates: An Overview

Drawing templates are required for single line diagrams and schematics. A drawing template specifies the page size and orientation of the drawing, and can include a border and title block. Several standard drawing templates are delivered with the software. You usually create a new drawing based on one of these existing templates. However, you can create new templates or customize existing templates to meet your particular needs. This is accomplished inside SmartPlant Electrical.

The default storage locations for single line diagram and schematic templates are specified in Options Manager. If you create a new template, you store it on the server in order for it to be available for drawing creation.

Also, the software comes with a set of files containing borders and title blocks that fit the delivered drawing templates. If you create a new template, you may also need to create a new title block that fits your template correctly and that contains the appropriate information. You can create title blocks in Catalog Manager or in a suitable graphics application such as SmartSketch.

Note

- The software embeds symbols that you insert in the template. For this reason, if you modify a border or title block symbol and you want the change to appear in all of your templates, you must reinsert the symbol in each template individually.

Related Topics
- *Create a Drawing Template*, page 265
- *Open a Drawing Template*, page 266

Create a Drawing Template

1. Click **File** > **New** > **Template**.

2. Select the new blank template and click **File** > **Sheet Setup**.

3. On the **Sheet Setup** dialog box, select the desired page size and orientation for the template from the list; for example, **A4 Wide**.

4. To add a border or title block, click **Edit** > **Insert** > **Symbol**.

   💡 **Tips**

   - You must create the border or title block file in advance with suitable dimensions for the template using Catalog Manager or a suitable graphics application such as SmartSketch.

   - The inserted symbol is embedded in the template. For this reason, if you modify a border or title block symbol and you want the change to appear in all of your templates, you must reinsert the symbol in each template individually.

5. On the **Select Symbol File** dialog box, navigate to the desired file and click **Open** to display it in the drawing view.

6. To position the border on the template, do one of the following:

   - Press the **Esc** key to lock the border on to the template in the correct position relative to the origin.

   - Drag and drop the border in the position you require.

7. To add text to the template, click **Edit** > **Insert** > **Text**.

8. You can add other drawing objects in the appropriate file format or text if required by clicking **Edit** > **Insert** > **Symbol**.

9. Click **File** > **Save** and type a name for the file in the desired location.

   📝 **Note**

   - Drawing templates are saved in a special format that is recognized only in SmartPlant Electrical. Files in the format have a .spe extension, and you can only create them in SmartPlant Electrical.

Related Topics
- *Working with Drawing Templates: An Overview*, page 264

Open a Drawing Template

1. Click **File** > **Open** > **Template**.

2. Navigate to the location where your templates are stored and select the desired template.

3. To edit the template, click **File** > **Sheet Setup**.

4. On the **Sheet Setup** dialog box, select the desired page size and orientation for the template from the list; for example, **A4 Wide**.

5. To add a symbol to the template, click **Edit** > **Insert** > **Symbol**.

   ### 💡 Tip

   - You must create the drawing object in advance using Catalog Manager or a suitable graphics application such as SmartSketch.

6. On the **Select Symbol File** dialog box, navigate to the desired file and click **Open** to display it in the drawing view.

7. Move the drawing object to the desired position on the template.

8. To add text to the template, click **Edit** > **Insert** > **Text**.

9. Add or delete other drawing objects and text as required.

10. Click **File** > **Save As** and choose a name and location for the file.

Related Topics
- *Working with Drawing Templates: An Overview*, page 264

Title Block Macros

You specify macros in your title block by adding SmartText from the Catalog Manager and defining the appropriate macro in the user properties. For all the available properties in each macro category, you should refer to the Data Dictionary. This topic describes a number of examples of macros that you can use in SmartPlant Electrical drawing title blocks.

Plant Group Macros

The following macros specify the plant group properties in your title blocks. You can specify any plant hierarchy level according to the settings of your project that you defined in the Data Dictionary. Plant hierarchy macros use the syntax [Plant Group Name].[Property].[Level], where [Level] is a number that specifies the level of the plant group in the hierarchy. The following examples illustrate the use of this syntax for the **name** property:

Macro	Notes
&Plant.name	Highest level plant group does not need a number to designate the level
&Area.name.2	Area is under Plant in the plant hierarchy
&Unit.name.3	Unit is under Area in the plant hierarchy
&Subsystem.name.4	Subsystem is under Unit in the plant hierarchy

Document Property Macros

Document property macros use the syntax Document.[Property]. The following macros are available for specifying document properties:

Macro	Notes
&Document.ItemTag	Document drawing number as defined in the Data Dictionary
&Document.Description	Document description
&Document.LastRevision	The latest revision of a document
&Document.DocumentCategory	Document category as defined in the Data Dictionary
&Document.DocumentType	Document type as defined in the Data Dictionary
&Document.ItemStatus	Document status as defined in the Properties window for a document
&Document.YYY	User-defined field name where YYY is any other property defined in the Data Dictionary

Document Revision Macros

Document revision macros use the syntax Revision.[Property].[Sequence], where [Sequence] is a number that specifies the sequence of the revision. This way, you can arrange revisions in ascending or descending order in the title block. Use the following macros to specify the document revisions in your document title blocks:

Macro	Notes
&Revision.RevisionNumber.x	Revision number
&Revision.RevisionDate.x	Revision date
&Revision.CreateBy.x	Created by
&Revision.CheckBy.x	Checked by
&Revision.RevisionDescription.x	Revision description
&Revision.ZZZ.x	User-defined field name where ZZZ is any other property defined in the Data Dictionary

Macros for Multi-Sheet Schematics

Multi-sheet schematic macros include the total number of sheets in a schematic drawing, the sequence number of a particular sheet, and a sheet description. Use the following macros to specify properties of multiple sheets in the title blocks of your schematic drawings:

Macro	Notes
&Sheet.Quantity	Defines the total number of sheets in a multi-sheet schematic
&Sheet.Number	Defines the sequence number of a particular sheet
&Sheet.Description	Sheet description

Note

- When generating a report, for the correct macros to be retrieved on the title block of your drawing, your working environment must be the same as the drawing. For example, if the objects on your drawing were created in a section of your plant named Unit 1, you must be working in the Unit 1 environment when generating the report, otherwise the software can not retrieve the correct macros.

Related Topics
- *Create a Drawing Template*, page 265
- *Macro Definition Guidelines*, page 275
- *Working with Drawing Templates: An Overview*, page 264

Working with Schematics: An Overview

A schematic is a graphical representation of wiring and motor control systems. You create a schematic by assembling it from typical blocks.

A block is a graphical representation of an electrical item. Blocks are parts of a complete drawing. To create a block, start by creating an entire drawing in Catalog Manager or any CAD application such as SmartSketch, AutoCAD, or MicroStation. Once the complete drawing is ready, you select a part of the drawing and turn it into a block.

You can generate schematics for the following item types:

- All loads
- All converting equipment
- All disconnect equipment
- Circuits
- Buses
- Generators
- Battery banks

The software can generate a schematic by retrieving data for power related items from one block or several blocks. If your generated schematic drawings contain the same information (that is, the information repeats itself in identical sections of different schematic drawings), you should consider creating a schematic block for the repeated parts so that this single block can be used by several other schematic drawings. Note that you can create schematic drawings that include several sheets. Multi-sheet schematics are mostly used when you need to display complex electrical equipment for which a single sheet is not enough to show all the relevant wiring details of the equipment.

After creating a block, you define the macros for that block. A macro is the smart text that you attach to the block graphic. The software retrieves this smart text from the database and attaches it to the block, thus making it part of the block file. You can create your macros in Catalog Manager, SmartSketch, AutoCAD, or MicroStation.

Also, you must attach your blocks to templates. A template contains the definition for the desired title block, border, and page size. You attach your blocks to templates in Catalog Manager, SmartSketch, or any other CAD application.

Before generating a schematic drawing, you create a typical block and a typical schematic that you will attach to a load for which you want to generate a schematic drawing. For a detailed workflow for generating and using schematic drawings, see *Recommended Workflow for Generating Schematic Drawings*, page 271.

Related Topics
- *Create Schematic Blocks in Catalog Manager*, page 290
- *Create Schematic Blocks in SmartSketch*, page 291
- *Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation*, page 292
- *Macro Definition Guidelines*, page 275
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with the Elements of a Schematic Drawing*, page 273

Recommended Workflow for Generating Schematic Drawings

The following is a recommended workflow for generating schematics.

1. Create the graphical block files

Create a complete drawing that includes all the needed graphical entities. You will then use this complete drawing to create separate graphical blocks. Make sure that the complete drawing includes a title block. Define the smart text labels (macros) and place them next to the specified objects of the drawing. Create the individual schematic blocks and set their X and Y displacement coordinates from the page origin. You can use Catalog Manager or SmartSketch to create your schematic blocks, that is, symbols (files with the .sym extension). Also, You can use any other CAD application such as AutoCAD or MicroStation to create .dwg or .dgn files. For more information, see:

- *Create Schematic Blocks in Catalog Manager*, page 290

- *Create Schematic Blocks in SmartSketch*, page 291

- *Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation*, page 292

- *Macro Definition Guidelines*, page 275

2. Create and define a new typical block

In the **Reference Data Explorer**, create a new typical block. Edit the properties of the new typical block and associate it with a file that contains the graphical representation of the current typical block. For more information, see *Create Typical Schematic Blocks*, page 69.

3. Create and define a new typical schematic

In the **Reference Data Explorer**, create a new typical schematic. Edit the properties of the new typical schematic and select the typical blocks that will comprise the typical schematic. Define the displacement from the page origin for each block and set the block sequence if two or more blocks are associated with the same block type. Also, select a suitable template. For more information, see *Create a Typical Schematic*, page 71.

4. Create and define a new profile

In the **Reference Data Explorer**, create a new profile for the pertinent item type. Edit the properties of the new profile and associate it with a typical schematic. For more information, see *Create a Profile*, page 61.

5. Apply the new profile to a selected electrical item

Select an electrical item in the **Electrical Index** and use the **Apply Options** feature to apply the new profile to the selected load. For more information, see *Apply a Profile to an Item*, page 147.

6. Generate a schematic for the selected electrical item

Select an electrical item in the **Electrical Index** and generate the new schematic. For more information, see *Generate a Schematic*, page 282. For information about creating multi-item schematics for more than one tag, see *Generate a Multi-Item Schematic*, page 283.

📝 Notes

- As an alternative to steps 4, 5, and 6, you can select an electrical item in the **Electrical Index** and open the **Generate Schematic** dialog box where you can select a typical schematic and then generate the drawing. If the selected typical schematic contains more than one block of the same item type for which you have not predefined a sequence, you can now associate each point with its proper project item.

- Sometimes the software generates a schematic block with a scaling incompatibility between the schematic blocks and the template used to generate the schematic drawing. This scaling incompatibility occurs if you have used AutoCAD or MicroStation to create your graphical block files. To correct this problem, you have to set the units of measure (inches or millimeters) for the generated schematic drawing. For more information, see *Set Units of Measure for Generated Schematic Drawings*, page 284.

Related Topics
- *Create Schematic Blocks in Catalog Manager*, page 290
- *Create Schematic Blocks in SmartSketch*, page 291
- *Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation*, page 292
- *Macro Definition Guidelines*, page 275
- *Working with Schematics: An Overview*, page 270
- *Working with the Elements of a Schematic Drawing*, page 273

Working with the Elements of a Schematic Drawing

The software allows you perform various actions with an open schematic drawing. Apart from regular actions in a drawing such as zooming, selecting, saving, and so forth, you can do the following:

Option	Description
Moving items	Click an item to select it and then drag it to another place in the drawing. Save the schematic to keep the items you moved in their new places. Note that this does not affect the data in your database.
Inserting annotations and redlining	Add text labels, symbols, or watermarks to your schematic as annotations. Add text, linked symbols, circles, rectangles, and lines to your schematic as redlining. For more information, see *Managing Annotations and Redlining: An Overview*, page 356.
Renaming a document and entering revisions	Open your schematic and on the **Edit** menu, click **Document Properties**.
Saving a schematic as an external file	With an open schematic, on the **File** menu, click **Save As**. For more information, see *Save a Schematic as an External Document*, page 289.
Printing a schematic	On the **File** menu, click **Print**.

Related Topics
- *Managing Annotations and Redlining: An Overview*, page 356
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with Schematics: An Overview*, page 270

Macro Definition Guidelines

You can associate a schematic block with the following item types:

- Loads
- Converting equipment
- Disconnect equipment
- Circuits
- Buses
- Generators
- Battery banks

You place macros on each block to retrieve relevant information from the database. Also, macros define default values that appear in the drawing if no data for a particular property is found in the database. Macros are text labels with a preceding ampersand (&). You can define macros in Catalog Manager or one of the following CAD applications: SmartSketch, AutoCAD, or MicroStation.

Macros Defined in Catalog Manager

In Catalog Manager, you define each block as a symbol (.sym) file. In addition to using text labels as macros, you can also place SmartText labels in a symbol. The advantage of using a SmartText label to define a macro is that instead of having to remember the item type property and typing it, you can select the property you require from a list. However, you can only use SmartText labels to specify properties of the specific electrical item that you specified for the symbol. For example, if the symbol was created for a cable, you can only select cable properties; if you want to retrieve properties for an associated item such as a motor, you must use a text label for the macro.

Note that for all SmartText labels used in a symbol, you have to set the **Unit Space** property to **World**.

Macro Syntax

The macro text has to contain the SmartPlant Electrical item name, the name of the desired item property, and a sequence number, according to the syntax:

&[ItemType].[Property].[Sequence]

For example, if you want to specify a name of a motor, the text label has to be:

&Motor.ItemTag.1

If a particular block contains several items of the same item type (that is, several cables, control stations, and so on), you have to use the sequence number to ensure item uniqueness. For more details, see *Sequencing Items in Block Macros*, page 279.

Panel Macros

Use the following syntax to create macros for local panels or junction boxes that include circuits:

&Circuit.Panel.[Property].[Sequence]

Cable Macros

Use the following syntax to create macros for cables:

- &Cable.[Property].[Sequence] - for power cables
- &ControlCable.[Property].[Sequence] - for control cables
- &GroundingCable.[Property].[Sequence] - for grounding cables
- &InstrumentationCable.[Property].[Sequence] - for instrumentation cables

Supported Electrical Items

Each drawing block can support the following electrical items and their properties:

Loads

Load blocks can include all the items that appear connected between a particular load and its feeder circuit in the **Electrical Engineer** hierarchy. Also, load blocks include load-related object properties regardless of whether these objects appear in the **Electrical Engineer** or not:

- Direct load properties.
- Related power cable properties.
- Related grounding cable properties.
- Related control cable properties.
- Related instrumentation cable properties.
- Related feeder circuit properties.
- Related circuit internal properties (disconnect equipment)
- Related bus properties.
- Related cell properties.
- Related power distribution board properties.
- Related control station properties.
- Related I/O symbol item properties.

Circuits

- Direct circuit properties.
- Related circuit component properties (circuit breakers, fuses, and so on.). If data for more than one item of the same type (for example, two circuit breakers) has to be retrieved, the macro name ends with a sequence number such as "& CircuitBreaker.ItemTag.1".
- Related electrical parent item properties: circuit - bus, circuit - power distribution board, circuit - cell, and so on.
- Related I/O symbol item properties.

Control Stations

- Direct control station properties.

- Related control cable properties. If data for more than one cable has to be retrieved, the macro name ends with a sequence number, for example "&Cable.ItemTag.1".

- Related I/O symbol item properties.

Converting Equipment

- The properties of all electrical items connected above the converting equipment item up to the first feeder circuit and then down up to the first incomer circuit or load.

- Related I/O symbol item properties.

Power Source Items, Generators and Battery Banks

- The properties of all electrical items connected below the power source item up to the first incomer circuit.

- Related I/O symbol item properties.

Related Topics
- *Generate a Schematic*, page 282
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Sequencing Items in Block Macros*, page 279
- *Working with Schematics: An Overview*, page 270

Sequencing Items in Block Macros

You use a sequence number if a particular item type appears more than once in a specific block. The sequence numbers define the item uniqueness and the order in which the items appear in the generated schematic drawing. The sole function of the sequence numbers is to help the software identify the project items and retrieve the appropriate data from the right tags.

The sequence numbers are unique per item type. You add these sequence numbers to the macros so that the software can retrieve the correct data belonging to the appropriate project item. You sequence the items belonging to the same type in the ascending order starting from the first item that appears right before the load and continue up to the feeder circuit. These numbers represent the order in which the various items are connected in the **Electrical Engineer** or circuit internals in the **Electrical Index**. Note that these sequence numbers do not correspond or relate to the **Sequence in group** values entered in the **Properties** window.

Also, the sequence numbers do not have to be consecutive (for example, 1,2,3), but they have to be in ascending order, for example 1, 5, 12, 20 and so forth.

In order to match the macros with the appropriate project items, the software creates lists of project item types. These lists contain items that are ordered according to their electrical connections in the **Electrical Engineer** starting from the load and up to the feeder circuit. The lists also include all the associated internals that exist in the circuits (either panel or feeder circuits) that may exist along the electrical path. Once the lists are complete, the software matches each project item on the list with the macro that contains the appropriate sequence number. The software matches the first project item on the list with the macro that has the lowest sequence number then with the macro that has the next number, and so on.

Handling Load Feeders with Parallel Branches

If there are several electrical paths/branches from a load to its feeder (for example, forward/reverse power paths), the software starts creating the lists starting with the branch that has the lowest sequence number. You define the branch order by setting the value in the **Sequence in group** property of the item closest to the load of the pertinent branch.

After completing the list of the items of the first branch (including the feeder and its internals), the software goes on to list the items of the next branch. The next branch is the one that has the next numeric value in the **Sequence in group** property of the item closest to the load in that branch. Note that the second list does not include the circuit and its internals as they are already included in the first list.

Handling Incomer and Feeder Circuit Internals

Make sure that you define the sequence numbering of circuit internals in the following manner:

- For incomer circuit internals and riser circuit internals - descending order, starting with the item closest to the bus.

- For feeder circuit internals and coupler circuit internals - ascending order, starting with the item closest to the bus.

Handling Converting Equipment Items

The software handles converting equipment macros starting with the items that appear at the bottom of the schematic based on the design in the **Electrical Engineer**. For a transformer that has cables and circuits connected to both primary and secondary windings, the software first resolves the items that have the lowest sequence in group numbers. After resolving the items that are connected to the first secondary winding, the software continues resolving the items for the primary winding from the bottom of the schematic up to the feeding bus. Then, the software returns to the bottom of the schematic and continues resolving the items that are connected to the other secondary winding (if one exists).

Handling Non-Power Related Items of Schematic Drawings

After completing the list of the items along all the existing electrical paths from the load to the feeder, the software looks for the associated control stations. If several control stations are associated with the load, the **Sequence in group** value of the controls stations determines the macro that is associated with that control station.

Related Topics

Buses with Several Circuits - Schematic Generation Prerequisites

When generating a schematic for a bus with several circuits, the software retrieves the data correctly after you fulfill the following prerequisites:

1. The block sequence must follow the same ascending order as the sequence in group of the circuits for which you are generating a schematic.

2. The block that corresponds to the first circuit (that is, the circuit with the lowest sequence number in group) must always have block sequence 1 (one).

3. The block sequence numbers must always be consecutive, unless:

   - the schematic does not include the first circuit; in this case, the block sequence has to start with 2 (two).

   - the schematic does not include one or more circuits between the first and the last circuits in the group; in this case, the block sequence value for that circuit is skipped.

✎ Note

- When generating a schematic for a bus, the software retrieves the following data:

- Complete power path data of each circuit.

- All circuit-related items, such as control cables and signal runs.

Related Topics
- *Generate a Schematic*, page 282
- *Macro Definition Guidelines*, page 275
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with Schematics: An Overview*, page 270

Generate a Schematic

1. In the **Electrical Index**, expand the **Electrical Equipment** folder and select an item for which you want to generate a schematic.

   💡 **Tip**

   - The software allows you to generate a schematic for the following item types:

     - All loads

     - All converting equipment

     - All disconnect equipment

     - Circuits

     - Buses

     - Generators

     - Battery banks

2. Do one of the following:

   - Click **Actions** > **Generate Schematic**.

   - Right-click the selected item and then click **Generate Schematic**.

   - Click 🔲 on the toolbar.

3. On the **Generate Schematic** dialog box, under **Typical schematic**, select an appropriate typical schematic that you created in the **Reference Data Explorer**.

4. Under **Drawing name**, type a name for the new schematic.

   💡 **Tip**

   - Examine the block data in the data window. Make sure that the typical blocks are defined correctly and their displacement from the page origin is set correctly.

5. Click the **Macros** tab. Examine the macros in the data window. Make sure that each macro is associated with a correct item. To associate a macro with another item, select a macro in the data window and then under **Macro details**, select another item from the **Macro item name** list.

6. Click the **Unresolved Items** tab to check whether there are item tags for which there are no corresponding items in the associated typical schematic.

7. Click **OK** to generate the schematic.

   📝 **Note**

   - The software can only generate schematic drawings for items that have pre-defined typical schematics.

Generate a Multi-Item Schematic

1. Create symbol (.sym) files for the following:

   - Multi-tag header for the main drawing page.

   - Multi-tag header for subsequent pages (usually the same as the multi-tag header for the main drawing page, but placed at the center of the drawing sheet).

   - Multi-tag macros.

   💡 **Tips**

   - Ensure that the column widths, the number of columns, and the properties that the macros represent match the corresponding characteristics of the headers.

   - Since at this stage, the macros are not associated with a specific item type, the syntax of the macro needs to include an expression representing the block type to which the item type refers, as follows: [ItemType].[Property].[Sequence].[BlockType].[BlockSequence]

     For example:

     &Cable.ActualLength.1.Motor.1
     &Cable.ActualLength.2.Motor.1
     &ControlStation.ItemTag.1.ControlStation.1
     &ControlStation.ItemTag.1.ControlStation.2

   - Define a typical block for each of the symbol files that you created using the following block types:

     - Multi-tag Header on Drawing Page.

     - Multi-tag Header.

     - Multi-tag Macros.

   - Define typical blocks for each block type that has a reference in the macros, for example, Motor, Control Station, Circuit.

   - In the Reference Data Explorer, create a typical schematic that includes all the typical blocks that you defined in the previous steps.

   - In the **Typical Schematic Common Properties** dialog box, select a multi-tag block type and click **Multi-Tag Options**.

   - On the **Multi-Tag Schematic Options** dialog box, select whether to include a list on the main drawing sheet, and specify the numbers of rows of data to appear on the main and subsequent drawing sheets.

- Apply this typical schematic to the tags that you want to appear in the multi-item schematic.

  💡 **Tip**

  - The tags must belong to the same subclass, for example, motors, heaters, or transformers, and so forth.

- In the Electrical Index, expand the **Documents** folder, and right-click the **Schematic Drawings** folder.

- Click **New Multi-Tag Schematic**.

- Right-click the newly-created schematic, and on the shortcut menu, click **Document Properties**.

- On the **Document Properties** dialog box, from the **Multi-tag typical schematic** list, select the typical schematic that you created in step 4.

- Click the **Associated Items** tab, and click **Associate**.

- In the **Find** dialog box that opens, under **Equipment type**, select the item type for which you want to associate tags, for example, **Motor**, and then click **Find Now**.

  💡 **Tip**

  - Only those tags to which you applied the selected typical schematic appear in the **Results** list.

- From the **Results** list, select the items that you want to associate with the multi-item schematic, and then click **OK**.

- Do one of the following to generate the schematic:

  - Right-click the schematic, and on the shortcut menu, click **Open**.

  - Right-click any of the associated items, and on the shortcut menu, click **Generate Schematic**. Ensure that in the dialog box that opens, under **Drawing name**, you select the appropriate multi-item drawing.

📝 **Note**

- When saving a multi-tag schematic as an external file, you need to save each page of the schematic as a separate drawing.

Set Units of Measure for Generated Schematic Drawings

1. With the schematic drawing open, expand the folder hierarchy **Tools** > **Options**.

2. On the **Options** dialog box, click the **Foreign Data** tab.

3. Under **Format**, click **MicroStation** or **AutoCAD** depending on the application that you used to create graphical block files.

4. Under **Import**, select the desired unit of measure from the **File units** list.

5. Click **OK**.

6. Close the schematic drawing.

7. Regenerate the schematic drawing.

Related Topics

Including Enhanced SmartLoop Reports with Schematics

When generating a schematic drawing for an electrical item, you can also specify the display of an Enhanced SmartLoop report containing loop wiring data, provided the signal for the item for which you display the schematic is associated with a loop. The loop data comes from SmartPlant Electrical after you publish the signal data to SmartPlant Instrumentation, make the appropriate wiring and loop associations, and publish the data back to SmartPlant Electrical.

Related Topics
- *Create Schematic Blocks in Catalog Manager*, page 290
- *Create Schematic Blocks in SmartSketch*, page 291
- *Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation*, page 292
- *Macro Definition Guidelines*, page 275
- *Working with Schematics: An Overview*, page 270
- *Working with the Elements of a Schematic Drawing*, page 273

Include Enhanced SmartLoop Reports with Schematics

1. In SmartPlant Electrical Options Manager, under **General Settings**, do the following:

   - Beside **Include Enhanced Reports in Schematics**, select **Yes**.

   - Beside **Default SmartPlant Instrumentation Domain Name**, type the name of the source domain.

2. In SmartPlant Electrical, click **File** > **Open** > **Template**.

3. Select the template that you want to use for displaying the Enhanced SmartLoop report.

4. Click **View** > **Display** > **Working Area**.

5. On the template drawing sheet, use the grab handles to define the size of the working area in which the Enhanced SmartLoop report is to appear.

6. Save the template.

7. For the item for which you intend to generate the schematic, publish signal data to SmartPlant Instrumentation.

   💡 **Tip**

   - The item for which you want to include the Enhanced SmartLoop report must have a related signal.

8. In SmartPlant Instrumentation, create loops and associate them with the retrieved signal data.

9. Publish the data back to SmartPlant Electrical, and generate the schematic for the item using the template that you modified.

   📝 **Notes**

   - The software displays the schematic on the initial drawing sheets and Enhanced SmartLoop reports on subsequent drawing sheets, with one or more sheets per loop.

   - The software supports multi-sheet Enhanced SmartLoop reports.

   Related Topics
   - *Generate a Schematic*, page 282
   - *Including Enhanced SmartLoop Reports with Schematics*, page 286
   - *Prerequisites for Working with the SmartPlant Instrumentation Interface*, page 449
   - *Recommended Workflow for Generating Schematic Drawings*, page 271
   - *SmartPlant Instrumentation Interface: An Overview*, page 448
   - *Working with Schematics: An Overview*, page 270

Generate Schematics in Batch Mode

1. In the **Electrical Index**, expand the **Electrical Equipment** folder that contains items for which you want to generate a schematic.

   💡 **Tip**

   - The software allows you to generate a schematic for the following item types:

     - All loads

     - All converting equipment

     - All disconnect equipment

     - Circuits

2. In the list view, select several items.

3. Click **Actions > Generate Schematic**.

4. On the **Batch Options for Schematics** dialog box, select the **Save files to disk** check box if you want to save the generated schematics as files.

5. Under **Path**, type the exact path where the files will be saved, or click the ellipsis button ⬚ to browse to the desired folder.

   💡 **Tip**

   • You can accept the displayed default path which you set in Options Manager.

6. From the **Save as type** list, select a file format.

7. Select the **Print drawings** check box if want to print out the generated schematics.

8. Click **OK** to generate the schematics.

   📝 **Note**

   • The software can only generate schematic drawings for items that have pre-defined typical schematics.

Related Topics
   • *Generate a Schematic*, page 282
   • *Open a Schematic Drawing*, page 288
   • *Recommended Workflow for Generating Schematic Drawings*, page 271
   • *Save a Schematic as an External Document*, page 289
   • *Working with Schematics: An Overview*, page 270

Open a Schematic Drawing

1. In the **Electrical Index**, expand the hierarchy **Documents** > **Schematic Drawings**.

2. Select a schematic drawing.

3. Click **File** > **Open** > **Drawing**.

   💡 **Tip**

   • Also, you can right-click a schematic drawing and on the shortcut menu, click **Open**.

Related Topics
   • *Generate a Schematic*, page 282
   • *Generate Schematics in Batch Mode*, page 287
   • *Recommended Workflow for Generating Schematic Drawings*, page 271
   • *Working with Schematics: An Overview*, page 270

Save a Schematic as an External Document

1. With the schematic drawing open, click, **File** > **Save As**.

2. On the **Save As** dialog box, navigate to the desired folder.

3. Type the file name and select the desired file type.

4. Click **OK**.

📝 Note

- If the schematic contains annotations, save it as an .spe file.

Related Topics

- *Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation*, page 292
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with Schematics: An Overview*, page 270

Create Schematic Blocks in Catalog Manager

1. Open Catalog Manager.

2. Select the symbol folder in which you want to store the new symbol.

3. Create a new symbol in the **Catalog**.

4. Open the symbol drawing and set the Default layer to active.

5. Select a suitable sheet size for your drawing.

💡 Tips

- Use the **Grid Display** and **Grid Snap** commands to place the origin properly.

- Edit the origin properties and set the origin layer as **hidden objects**.

6. Draw your block. Make sure that you make the **Default** layer as the active layer.

7. Add the SmartText.

💡 Tips

- If the SmartText requires repositioning, change the active layer to **Label**.

- Do not forget to change the layer back to **Default** when you return to the drawing.

8. Add the macro text to retrieve the attributes that are not available for SmartText. For more information about macro text, see *Macro Definition Guidelines*, page 275.

9. Save the graphic as a symbol (.sym) file.

10. If the new symbol file is a complete typical schematic drawing, use it to create individual schematic blocks. For more information, see *Use a Complete Symbol File to Create Schematic Blocks in Catalog Manager*, page 290.

Related Topics
- *Generate a Schematic*, page 282
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Use a Complete Symbol File to Create Schematic Blocks in Catalog Manager*, page 290
- *Working with Schematics: An Overview*, page 270

Use a Complete Symbol File to Create Schematic Blocks in Catalog Manager

1. Open the complete schematic symbol file. (See step 9 in *Create Schematic Blocks in Catalog Manager*, page 290.)

2. Delete all the graphics, SmartText, and text labels that are not part of the desired drawing block.

3. Save the remaining graphics, SmartText, and text labels to a new symbol file. Use the **Save As** command.

4. Repeat steps 1 through 3 to create the rest of the schematic drawing blocks.

Related Topics
- *Create Schematic Blocks in Catalog Manager*, page 290
- *Generate a Schematic*, page 282
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with Schematics: An Overview*, page 270

Create Schematic Blocks in SmartSketch

1. Open SmartSketch.

2. Create a new drawing file (.igr) or open an existing schematic drawing.

    💡 **Tip**

    *   You can open CAD files created in AutoCAD (.dwg) or MicroStation (.dgn).

3. Set the following parameters for an open drawing:

    *   Set all the layers in use to visible.

    *   Set the Default layer as the active layer.

    *   Move all the graphic objects from their original layer to the Default layer.

    *   Select a suitable sheet size for your drawing.

    *   Move the drawing origin to the bottom left corner of the sheet (0.0).

    *   Add all the macros for the desired entities by defining the text labels starting with an ampersand.

4. Save the .igr drawing.

5. Open the drawing and set the Default and Label layers to visible. Hide all the other layers.

6. If the .igr file is a complete typical schematic drawing, use it to create individual schematic blocks by doing the following:

    *   Select a set of graphics and macros that will comprise the new block symbol.

    *   Click the **Create Symbol** icon.

    *   Move the symbol origin to the bottom left corner of the drawing sheet.

    *   Save the new drawing as a symbol file in your default symbol folder.

Related Topics

Guidelines for Creating Schematic Blocks in AutoCAD and MicroStation

1. There is no need to create blocks or cells. SmartPlant Electrical recognizes .dwg and .dgn files and treats each drawing as a SmartPlant Electrical block file.

2. Set the drawing sheet origin at the bottom left corner of the drawing sheet (0.0).

Related Topics
- *Generate a Schematic*, page 282
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Set Units of Measure for Generated Schematic Drawings*, page 284
- *Working with Schematics: An Overview*, page 270

Working with Single Line Diagrams: An Overview

SmartPlant Electrical enables you to generate the following types of single line diagrams:

- A power distribution board-based single line diagram

- An **Electrical Engineer**-based single line diagram

- A blank single line diagram template

A PDB-based single line diagram (SLD) is a document that includes a power distribution board general arrangement drawing and a power distribution board (PDB) schedule. The software generates a single line diagram for a selected power distribution board that contains all the PDB buses, circuits, circuit components, and their inter-connections (bus couplings).

A single line diagram that you generate from the **Electrical Engineer** is a main or key single line diagram of the electrical network for a selected set of items created according to the generation options that you predefine.

A blank template allows you perform a drag-and-drop operation from the **Electrical Index** on those items that you want to appear in your single line diagram. These items include buses, offsite power supplies, generators, battery banks, converting equipment not contained within PDBs, and standalone disconnect electrical equipment. After positioning the items on the SLD template, the software will complete the drawing according to the generation options that you defined.

When generating an SLD, the software searches for a symbol for each electrical item that is included in the SLD source. SmartPlant Electrical supplies you with a set of default symbols that you can use in your drawings, or you can create your own custom symbols in Catalog Manager. After creating your symbols and defining them in the Options Manager, the software then uses the SLD symbols according to the following rules:

1. SmartPlant Electrical checks whether the items included in the SLD source are associated with symbol files. Wherever such an association exists, the software uses the associated symbol file to represent the electrical item in the single line diagram. You associate an item with a symbol file using the **Associate Custom Symbols** command from the **Actions** menu for an item that you select in the **Electrical Index**.

2. If SmartPlant Electrical cannot find a symbol file associated with a particular electrical item, the software uses a symbol with a name that matches the item type name of the electrical item to be included in the single line diagram. This symbol resides in the folder defined in the **Default SLD Symbol Path** location in Options Manager.

For more information about SLD symbols, see *Guidelines for Creating SLD Symbols in Catalog Manager*, page 315.

Related Topics
- *Working with Single Line Diagrams Common Tasks*, page 294

Working with Single Line Diagrams Common Tasks

The following tasks are used frequently when you generate and work with single line diagrams.

Generate a Single Line Diagram for a PDB

This option allows you to create a single line diagram for a PDB that you select in the **Electrical Index**. You start by selecting the PDB for which you want to generate an SLD. Then, you define the generation options such as the page settings and the settings for the display of the circuits and circuit components. After generating the SLD, you can save the SLD as an internal drawing in the software. Note that you can generate only one SLD per PDB. If you select a PDB for which you have previously created an SLD, the software opens the existing SLD using the generation options that you defined before the previous generation. To generate a completely new SLD for such a PDB, first delete the SLD from the **Documents** folder in the **Electrical Index** and then generate the SLD. For more information, see *Generate a Single Line Diagram for a PDB*, page 301.

Generate Multiple SLDs in Batch Mode

This option allows you to generate multiple single line diagrams for a PDB in batch mode. You can use this option to print the generated SLDs right after the software generates them with or without saving the SLDs as external files. For more information, see *Generate Multiple Single Line Diagrams for PDBs in Batch Mode*, page 303.

Generate an SLD from the Electrical Engineer

This option allows you to generate a single line diagram based on the items you select in the **Electrical Engineer**. First, you select the items from which you want the new SLD to originate. Then, you set the generation options to define the electrical items that serve as the initial points or sources of your SLD, define the SLD page and display settings, as well as the definitions for the display of circuits and circuit components. After generating the SLD, you can save the SLD as an internal drawing in the system or as an external drawing using the **Save As** command. You can generate as many SLDs for the same item as needed. For more information, see *Generate an SLD from the Electrical Engineer*, page 300.

Generate an SLD from a Blank SLD Template

This option allows you to generate a single line diagram using a blank template. First you drag and drop the items you want in your SLD onto the template, positioning them were you want them to be. Then, you set the generation options to define the electrical items that serve as the initial points or sources of your SLD, define the SLD page and display settings, as well as the definitions for the display of circuits and circuit components. After generating the SLD, you can save the SLD as an internal drawing in the system or as an external drawing using the **Save As** command. You can generate as many SLD for the same item as needed. If you want to regenerate an open SLD drawing using different generation options, with the SLD open, click **Tools > Drawing Options**. For more information, see *Generate an SLD from a Blank SLD Template*, page 303.

Remove an Item from an SLD

This topic explains how to remove an item from a Single Line Drawing that was generated using a blank template. For more information, see *Remove an Item from an SLD*, page 311.

Generate an SLD with Bus Tie and Internals

This topic explains how to show the internals of a bus tie in a horizontal orientation on an SLD. For more information, see *Generate an SLD with Bus Tie and Internals*, page 304.

Opening Single Line Diagrams

This topic explains how to open single line diagrams that you have saved as internal drawings. For more information, see *Opening Existing Single Line Diagrams: An Overview*, page 296.

Working with SLD Elements

This topic describes the various activities that you can perform with an open single line diagram. For more information, see *Working with SLD Elements*, page 299.

Find an Electrical Item in an SLD

This feature allows you to find an electrical item in an open single line diagram. You select an electrical item in the **Electrical Engineer** or the **Electrical Index** and the software finds the item in the open SLD. For more information, see *Find an Electrical Item in an SLD*, page 308.

Add or Insert a Legend

While defining SLD generation options, the software allows you to include a legend which explains the meaning of all the symbols used in the generated SLD. The software also allows you to insert a legend in an existing SLD that was generated without a legend. Once an SLD contains a legend, you can drag the legend and move it like any other element in an SLD. For more information, see *Insert a Legend in an Existing SLD*, page 309.

Show Space Heaters in an SLD

This option allows you to show space heaters adjacent to their associated motor on your SLD. For more information, see *Show Space Heaters in an SLD*, page 308.

Show Control Stations in an SLD

This procedure allows you to show control stations associated with equipment on your SLD. For more information, see *Show Control Stations in an SLD*, page 307.

Show Metering and Relaying Equipment in an SLD

This option allows you to show metering and relaying equipment associated with items on your SLD. For more information, see *Show Metering and Relaying Equipment in an SLD*, page 307.

Managing Annotations and Redlining

You can mark up an SLD with annotations or redlining. The software saves annotations in the same layer as the graphical elements of the SLD, whereas redlining appears in a separate drawing layer and you can hide and display redlining as you require. You can use text or symbols as annotations, and text, symbols, or graphics as redlining. For more information, see *Managing Annotations and Redlining*, page 357.

Add and Maintain SLD Revisions

You can add and maintain revisions in single line diagrams. The software allows you to select a revision numbering method, enter a revision number, date, short description, and the name of the person who modified the SLD. Also, you can add or update the last revision of a number of selected documents. For more information, see *Working with Internal Documents Common Tasks*, page 373.

Save a Single Line Diagram as an External File

This procedure explains how to save a single line diagram as an external file. For more information, see *Save a Single Line Diagram as an External File*, page 315.

Opening Existing Single Line Diagrams: An Overview

The software allows you to open single line diagrams that you generated previously. These drawings are single line diagrams that you saved as internal drawings in the software. If you have not changed any electrical items that pertain to the SLD that you want to open, the software displays the drawing as it was saved the last time. However, if you have modified some of the electrical data that pertains to that SLD, the software regenerates the drawing by retrieving the information from the database. The software keeps the previous generation options and all the existing annotations. The items that have not been modified retain their previous positions in the SLD and keeping their existing coordinates. The new or modified electrical items appear in the drawing with the coordinates that are calculated by the software.

You can open a single line diagram using one of the following methods:

- **SLDs for power distribution boards** - You open these single line diagrams from the **Electrical Index**. For more information, see *Open an SLD for a Power Distribution Board*, page 305.

- **SLDs based on items in the Electrical Engineer** - You open these single line diagrams from the **Electrical Engineer**. For more information, see *Open an SLD Based on an Electrical Engineer Item*, page 304.

- **SLDs in the Documents folder in the Electrical Index** - You can open an existing single line diagram directly without selecting the electrical item for which it was generated if you know the name of the SLD. The SLDs in the **Documents** folder are the single line diagrams that you saved as internal drawings. For more information, see *Open an SLD from the Documents Folder*, page 306.

Related Topics
- *Generate a Single Line Diagram for a PDB*, page 301
- *Generate an SLD from the Electrical Engineer*, page 300
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Working with SLD Elements

The software allows you perform various actions with an open single line diagram. Apart from regular actions in a drawing such as zooming, selecting, saving, and so forth, you can do the following:

Option	Description
Moving items	Click an item to select it and then drag it to another place in the drawing. Save the SLD to keep the items you moved in their new places. Note that this does not affect the data in your database.
Entering and modifying item properties	Select an electrical item and then enter or modify its properties in the **Properties** window.
Changing the line type, line color and line width of a selected bus	Select a bus and then click the appropriate icon on the SLD toolbar to change the line type, line width, or line color for the selected bus. Note that the software retains new line display settings only if you save the SLD as an external document. Otherwise, after reopening the SLD, the software displays the original bus symbol. You can use Options Manager to change the symbol that the software uses to represent buses. To create a new symbol for buses, use Catalog Manager. For more information, see *Guidelines for Creating SLD Symbols in Catalog Manager*, page 315.
Inserting annotations and redlining	Add text labels, symbols, or watermarks to your SLD as annotations. Add text, linked symbols, circles, rectangles, and lines to your SLD as redlining. For more information, see *Managing Annotations and Redlining: An Overview*, page 356.
Renaming a document and entering revisions	Open your SLD and on the **Edit** menu, click **Document Properties**.
Saving an SLD as an external file	With an open SLD, on the **File** menu, click **Save As**. For more information, see *Save a Single Line Diagram as an External File*, page 315.
Setting SLD generation preferences	On the **File** menu, click **Preferences**. For more information, see *Customize Preferences for SLD Generation*, page 115.
Setting new generation options and regenerating an SLD	On the **Tools** menu, click **Drawing Options**. For more information, see *Generate an SLD from the Electrical Engineer*, page 300.
Printing an SLD	On the **File** menu, click **Print**.

Note

- The software uses the following conventions to display parallel cables in an SLD:

  [...] in the item tag label and (xN) in the cable configuration label, where N is the number of cables, for example:

  Cable-32/A[...]

  3 x 150mm2 (x5)

  This means that there are five parallel cables represented by the same line in the SLD.

Generate an SLD from the Electrical Engineer

1. In the **Electrical Engineer**, select the items for which you want to generate your single line diagram.

2. Do one of the following to open the **Single Line Diagram Options** dialog box.

   - Click **File** > **New** > **SLD**.

   - Right-click a highlighted item and then on the shortcut menu, click **New SLD**.

3. On the **Single Line Diagram Options** dialog box, define the SLD generation options as you require.

4. Click **Generate**.

Notes

- If you want to regenerate an open SLD drawing using different generation options, with the SLD open, click **Tools** > **Drawing Options**.

- To save the drawing as an internal document, in the generated SLD drawing, click **File > Save**.

- If there are overlapping connectors on the generated drawing, open the SLD and then click **Tools > Optimize Overlapping Connectors** to reposition the connectors so that they do not overlap. Note that this operation may take a few moments if your SLD contains a lot of connectors.

- For information on how to use custom symbols in your SLDs and the SLD legends, see *Use Custom Symbols in SLDs*, page 309.

- If, during the SLD generation, the software cannot find a referenced symbol for a specific electrical item, the software uses the default symbol file (default.sym) to represent the item. You specify default files in Options Manager.

Related Topics

- *Working with Single Line Diagrams Common Tasks*, page 294

Generate a Single Line Diagram for a PDB

1. In the **Electrical Index**, select the power distribution board for which you want to generate a single line diagram.

2. Do one of the following to open the **Single Line Diagram Options** dialog box:

   - Click **Actions** > **Generate SLD for PDB**.

   - Click ⊞ on the toolbar.

   - Right-click a highlighted item and then on the shortcut menu, click **Generate SLD**.

3. On the **Single Line Diagram Options** dialog box, define the SLD generation options as you require.

4. Click **Generate**.

📝 **Notes**

   - To save the drawing as an internal document, in the generated SLD drawing, click **File > Save**.

   - If there are overlapping connectors on the generated drawing, open the SLD and then click **Tools > Optimize Overlapping Connectors** to reposition the connectors so that they do not overlap. Note that this operation may take a few moments if your SLD contains a lot of connectors.

   - For information on how to use custom symbols in your SLDs and the SLD legends, see *Use Custom Symbols in SLDs*, page 309.

   - If, during the SLD generation, the software cannot find a referenced symbol for a specific electrical item, the software uses the default symbol file (default.sym) to represent the item. You specify default files in Options Manager.

Related Topics
   - *Customize Preferences for SLD Generation*, page 115
   - *Generate an SLD from the Electrical Engineer*, page 300
   - *Generate Multiple Single Line Diagrams for PDBs in Batch Mode*, page 303
   - *Opening Existing Single Line Diagrams: An Overview*, page 296
   - *Use Custom Symbols in SLDs*, page 309
   - *Working with Single Line Diagrams Common Tasks*, page 294
   - *Working with SLD Elements*, page 299

Generate an SLD from a Blank SLD Template

1. On the **Electrical Index** open the **Documents** folder.

2. On the **Single Line Diagrams** folder, right-click and select **New Manual SLD**.

3. Select the new SLD in the **Single Line Diagrams** folder, and do one of the following:

   - Right-click, and from the shortcut menu select **Open**.

   - Click **File** > **Open** > **SLD**.

4. From the Electrical Index ,drag and drop onto the opened template the items you want in your SLD.

5. Click **Tools** > **Drawing Options**.

6. On the **Single Line Diagram Options** dialog box, define the options you require for your SLD template.

7. Click **Generate**.

📝 Note

   - If you want to regenerate an open SLD drawing using different generation options, with the SLD open, click **Tools** > **Drawing Options**.

Related Topics

   - *Working with Single Line Diagrams: An Overview*, page 293

Generate Multiple Single Line Diagrams for PDBs in Batch Mode

1. In the **Electrical Index**, select the **Power Distribution Equipment** folder and select **Power Distribution Boards**.

   ### 💡 Tip

   - You can also generate multiple SLDs in batch mode from the Tabular Editor.

2. In the list view pane of the **Electrical Index** select the PDBs you require and do one of the following to open the **Batch Options for SLDs** dialog box:

   - Click **Actions** > **Generate SLD for PDB**.

   - Right-click the selected items and then on the shortcut menu, click **Generate SLD**.

3. On the **Batch Options for SLDs** dialog box, do the following as you require:

   - To save the SLDs as external files, select **Save files**, enter the file path where you want to save the files, and then select the file type that you require.

   - To print the generated SLDs automatically right after the software generates them, select **Print drawings**.

📝 **Note**

The software generates the single line diagrams according to the following default settings:

   - The SLDs show all circuit internals.

   - The SLDs show cables and cable terminations.

   - The software uses sequence numbers for circuits.

   - The software uses the default SLD template that you defined in Options Manager.

   - The SLDs do not display a legend that explain the symbols used in the SLDs.

Related Topics

Generate an SLD with Bus Tie and Internal Components

1. On the **Electrical Index** create your PDB including the Bus Tie coupler-riser relations with their circuit internals.

2. From the **Electrical Engineer**, select the circuit with the Bus Tie coupler-riser that you want to create an SLD for, and do one of the following:

   - Click **File** > **New** > **SLD**.

   - Right-click on the circuit, and then on the shortcut menu, click **New SLD**.

   From the **Electrical Index**, select the PDB that contains the Bus Tie coupler-riser that you want to create an SLD for, and do one of the following:

   - Click **Actions** > **Generate SLD for PDB**.

   - Click 🔀 on the toolbar.

   - Right-click the PDB, and then on the shortcut menu, click **Generate SLD**.

3. On the **Single Line Diagram Options** dialog box, select the **Circuits and Internals** tab, and do the following:

   - Select the options you require.

   - Set the distance between the circuit internals and the circuits, remembering to make the spacing wide enough so that the circuit items will not be crowded onto each other, and require moving manually.

4. Select the **Buses** tab and select the required options.

5. Select the required options from the other tabs.

6. Click **Generate**.

Note

   - For you to see the Bus Tie and its internal circuit you must use the correct symbols. The Coupler circuit symbol, CircuitBusTieCoupler.sym and the Riser circuit symbol, CircuitBusTieRiser.sym. Any of the symbols used in the internal circuits must use the syntax **<itemtype>BusTie.sym**, for example CircuitbreakerBusTie.sym.

Related Topics
   - *Associate Bus Riser Circuits with Coupler Circuits*, page 226
   - *Working with Single Line Diagrams Common Tasks*, page 294

Open an SLD Based on an Electrical Engineer Item

1. In the **Electrical Engineer**, select the item for which you want to open a single line diagram.

2. Click **File** > **Open** > **SLD**.

Notes

   - You can right-click an item and click **Open SLD** on the shortcut menu.

   - If the electrical item you selected is an origin for more than one SLD drawing, the software opens the **Open Single Line Diagram** dialog box where you select the drawing that you want to open.

   - If you have opened an SLD drawing whose electrical data has changed (for example, you added or deleted a bus, moved a circuit, and so on), the software does not display the SLD clearly and some of the items might overlap. To fix the SLD, with the SLD drawing open, click **Tools > Drawing Options**. Set the required SLD options and click **Generate**. The regenerated SLD drawing retains the existing annotations. However, it does not keep the old circuit and circuit component placements in the drawing.

Open an SLD for a Power Distribution Board

1. In the **Electrical Index**, highlight a power distribution board.
2. Click **File** > **Open > Drawing**.

📝 **Notes**

- You can right-click an item and click **Open SLD** on the shortcut menu.

- If you have opened an SLD drawing for a PDB whose electrical data has changed (for example, you added or deleted a bus, moved a circuit, and so on), the software does not display the SLD clearly and some of the items might overlap. To fix the SLD, with the SLD drawing open, click **Tools > Drawing Options**. Set the desired SLD options and click **Generate**. The regenerated SLD drawing retains the existing annotations. However, it does not keep the old circuit and circuit component placements in the drawing.

Related Topics
- *Opening Existing Single Line Diagrams: An Overview*, page 296
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Open an SLD from the Documents Folder

1. In the **Electrical Index**, expand the following hierarchy: **Documents** > **Single Line Diagrams**.
2. Select an SLD.
3. Click **File** > **Open** > **Drawing**.

📝 **Notes**

- You can right-click an SLD and click **Open** on the shortcut menu.

- If you have opened an SLD drawing for which the electrical data has changed (for example, where you added or deleted a bus, moved a circuit, and so on), the software does not display the SLD clearly and some of the items might overlap. To fix the SLD, with the SLD drawing open, click **Tools > Drawing Options**. Set the required SLD options and click **Generate**. The regenerated SLD drawing retains the existing annotations. However, it does not keep the old circuit and circuit component placements in the drawing.

Related Topics
- *Opening Existing Single Line Diagrams: An Overview*, page 296
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Show Control Stations in an SLD

1. Open a single line diagram. For more information, see *Opening Existing Single Line Diagrams: An Overview*, page 296.

2. Click **Tools** > **Drawing Options**.

3. On the **Single Line Drawing Options** dialog box, click the **Related Equipment** tag.

4. Under **Control Station options,** check **Show control stations associated with equipment.**

5. Under **Control stations displacement,** type the values for the **X:** and **Y:** coordinates, and select the units required from the **Units** list.

6. Click **Generate**.

Related Topics
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with Single Line Diagrams: An Overview*, page 293

Show Metering and Relaying Equipment in an SLD

1. Open a single line diagram. For more information, see *Opening Existing Single Line Diagrams: An Overview*, page 296.

2. Click **Tools** > **Drawing Options**.

3. On the **Single Line Drawing Options** dialog box, click the **Related Equipment** tag.

4. Under **Metering & Relaying Equipment options,** check **Show metering and relaying equipment.**

5. Under **Metering equipment first symbol displacement,** type the values for the **X:** and **Y:** coordinates, select the units required from the **Units** list, and type the value for the **Vertical displacement**.

6. Click **Generate**.

🖉 **Note**

- The logical connections are completed by you, on the drawing, after the drawing has been generated.

Related Topics
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with Single Line Diagrams: An Overview*, page 293

Show Space Heaters in an SLD

1. Open a single line diagram. For more information, see *Opening Existing Single Line Diagrams: An Overview*, page 296.

2. Click **Tools** > **Drawing Options**.

3. On the **Single Line Drawing Options** dialog box, click the **Related Equipment** tag.

4. Under **Space heater options,** check **Show space heaters associated with motors.**

5. Under **Space heater symbol displacement,** type the values for the **X:** and **Y:** coordinates, and select the units required from the **Units** list.

6. Click **Generate**.

Related Topics
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with Single Line Diagrams: An Overview*, page 293

Find an Electrical Item in an SLD

1. Open or generate a single line diagram.

2. In the **Electrical Engineer** or **Electrical Index**, expand the pertinent hierarchy in the tree view.

3. Right-click an electrical item and select **Find in SLD** on the shortcut menu.

📝 Notes

- The software highlights the item it finds in the SLD. You can now click 🔍 to modify the magnification level of the selected item. To cancel the selection, click somewhere outside the drawing.

- In redlining mode, the **Find in SLD** option is not available.

Related Topics
- *Generate a Single Line Diagram for a PDB*, page 301
- *Opening Existing Single Line Diagrams: An Overview*, page 296
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Insert a Legend in an Existing SLD

1. Open a single line diagram. For more information, see *Opening Existing Single Line Diagrams: An Overview*, page 296.

2. Click **Edit > Insert > Legend**.

3. On the **SLD Legend** dialog box, select one of the following options:

   - **Horizontal**.

   - **Vertical**.

4. In the SLD, click where you want the software to insert the legend.

📝 Notes

- Once an SLD contains a legend, it behaves like any other element in the SLD. Therefore, you can resize it or drag it to another place in the SLD.

- The SLD legend contains the symbols that represent the electrical items in the open single line diagram. Also, the software shows custom symbols if you have defined the **Electrical Equipment Type** property and selected a symbol from the **Symbol Name** list in the **Properties** window for a particular electrical item.

- For information on how to use custom symbols in your SLDs and the SLD legends, see *Use Custom Symbols in SLDs*, page 309.

Use Custom Symbols in SLDs

1. In the **Electrical Index** or **Electrical Engineer**, select the item tag you want to associate with a custom symbol.

2. Do one of the following:

   - Click **Actions > Associate Custom Symbols.**

   - Click .

   - Right-click > **Associate Custom Symbols.**

3. On the **Associate Custom Symbols** dialog box, under **Document Type** click on a cell, and from the list select the document type you require.

4. Under **Symbol**, click .

5. Browse to the symbol (.sym) file you want to add, and select the file.

6. Click **OK**.

✎ Notes

- You can only choose symbols that are stored in the default folder, and it's sub-folders. You set the default symbol folder in the SmartPlant Electrical Options Manager. For more details, see Defining File Locations: An Overview in the *SmartPlant Electrical Option Manager Users Guide*.

- If, during the SLD generation, the software cannot find a referenced symbol for a specific electrical item, the software uses the default symbol file (default.sym) to represent the item. You specify default files in Options Manager.

- For guidelines about the creation of custom symbols in Catalog Manager, see *Guidelines for Creating SLD Symbols in Catalog Manager*, page 315.

Related Topics
- *Customize Preferences for SLD Generation*, page 115
- *Generate a Single Line Diagram for a PDB*, page 301
- *Generate an SLD from the Electrical Engineer*, page 300
- *Place Symbol Connect Points*, page 317
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Customize a PDB Border (Frame) on a SLD

1. Open Catalog Manager and create or edit the PDB border symbol. For more information, see Create a Symbol in the *Catalog Manager User's Guide*, under *Working with Symbols*, *Creating Symbols*, *Create a Symbol*.

2. Open Options Manager and define the symbology for the PDB border symbol created in Catalog Manager. For more information, see Define an Item Type Symbol in the *Options Manager User's Guide*, under *Using Options Manager*, *Defining Item Type Symbology*, *Define an Item Type Symbol*.

3. In SmartPlant Electrical associate your PDB border symbol with the PDB you require. For more information, see *Associate a Custom Symbol with an Item Tag*, page 351

Remove an Item from An SLD

🛑 Important

- This procedure only works for SLDs created using a blank SLD template. It can not be used on SLDs that were automatically created by the software.

1. On your open SLD select the item you want to delete.

2. Do one of the following:

   - Click ✎.

   - Click **Edit** > **Remove**.

   - Right-click, and from the shortcut menu, click **Remove**.

Related Topics
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with Single Line Diagrams: An Overview*, page 293

Creating and Using Symbol Assemblies for SLD Diagrams: An Overview

The following guidelines apply when creating assemblies:

General Guide Lines

An assembly is a group of symbols related to a main item with related item types. The following item types are used as placeholders for assemblies:

- Circuits / typical circuits

- Loads

- Conventional equipment

- Generators

After assembly generation, you can select an individual item within an assembly, and view the item properties and perform connections to auxiliary points.
After assembly generation you will not be able to break the graphics into individual symbols that can be moved. The assembly will move as one group.

Assembly Composition Guidelines

Circuit / typical circuits

- Circuit data (main item data).

- All internals (including metering equipment).

- All I/O associated signals.

- Cable item tags associated to the circuit.

- Schematic item tag.

Loads/converting equipment/generators

- Main item data (circuit information and properties).

- Associated control station and associated cable item tags.

- Associated instruments and their cable item tag.

- Motor space heater for motors.

- Schematics item tag.

- I/O signals.

- control or instrument cables item, tags (embedded cable symbols retrieve cable data without referring to their category).

Transformers and generators:

- Neutral grounding resistor data.

- Neutral grounding cable data (as property only).

- Neutral grounding metering equipment data.

Related Topics
- *Creating and Using Symbol Assemblies for SLD Diagrams*, page 312

Creating and Using Symbol Assemblies for SLD Diagrams

An assembly is a symbol comprising two symbols or more, where there is a
functional relationship between the item types the symbols represent. Assemblies are
created within SLD items.
Draw and create the symbol assembly in the Catalog Manager.
You can view the symbol assembly and generate the SLD in SmartPlant Electrical.
In this way you can use assemblies to fully illustrate complex items such as circuits
with their internals. When creating an assembly or combining two assemblies or
more, the items in the assemblies must have valid relations. For details, see *Creating
and Using Symbol Assemblies for SLD Diagrams: An Overview*, page 311.

Create and Use a Symbol Assembly

1. Create a new item or use an existing one.

2. To create an assembly, drag the desired symbols to the drawing sheet.

   💡 **Tip**

   - You can use **Smart Text Editor** to create a label if desired. For
     details see .

3. In the **Properties** window, select the appropriate item type from the list, and click
   💾

4. .

5. In the Option Manager, click **Option** > **Locations**.

6. On the **Locations** window, verify that the **Default SLD Symbol Path** location,
   matches the symbol location in the Catalog Manager.

7. In the Option Manager, you can define the item type symbology. For details see,

8. In SmartPlant Electrical index tree view pane, right click the predefined item and
   on the shortcut menu click **Associate Custom Symbol**.

9. On the **Associate Custom Symbol** dialog box, click **Add**.

10. From the **Document type** column, from the list, select the desired document type.

11. From the **Symbol** column, select the assembly symbol to associate with the
    document type.

12. In the Electrical Engineer tree view pane, right click the item and on the shortcut menu, click **New SLD**.

13. On the **Single Line Diagram Options** dialog box, click **Generate** to view the SLD containing the assembly.

💡 **Tips**

- The assembly item properties now appear in the **Properties** window in SmartPlant Electrical.

- In an assembly, items of the same type are sequenced in-group. It is possible to change the sequence but two items cannot have identical sequence values.

- It is possible to combine several assemblies into one.

Related Topics
- *Creating and Using Symbol Assemblies for SLD Diagrams: An Overview*, page 311
- *Generate an SLD from the Electrical Engineer*, page 300
- *Working with Single Line Diagrams: An Overview*, page 293

Gapping Lines: An Overview

A gap in a drawing is a condition that exists when two lines intersect graphically in the drawing but not physically in the plant. In order to portray this condition, you can choose that lines show a gap at these non-physical intersections. This can help the clarity of your drawing considerably.

You can choose when the software performs gapping on your drawing by using the **Gap Now** command.

Related Topics
- *Gap Now Command (Tools Menu)*, page 566
- *Perform Line Gapping in Your Drawing Now*, page 315

Perform Line Gapping in Your Drawing Now

- Click **Tools > Gap Now**.

Related Topics
- *Gapping Lines: An Overview*, page 313

Save a Single Line Diagram as an External File

1. Open or generate a single line diagram.
2. Click **File** > **Save As**.

Related Topics
- *Generate a Single Line Diagram for a PDB*, page 301
- *Opening Existing Single Line Diagrams: An Overview*, page 296
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with SLD Elements*, page 299

Guidelines for Creating SLD Symbols in Catalog Manager

SmartPlant Electrical uses symbols to generate single line diagrams. You create these symbols in Catalog Manager. You store the symbol files in the folder that you define in the Default SLD Symbol Path location in Options Manager.

General Guidelines

When generating a single line diagram, SmartPlant Electrical searches for the symbols that match the item type names. Therefore, all the symbol names must match the item type names of the electrical items that you want to include in the SLD. You can, however, save a symbol file under a different name and assign it to an electrical item using the **Associate Custom Symbols** command from the **Actions** menu. In this case, the software will use the associated symbol file to represent the electrical item in an SLD.

You have to set an item type for each symbol that you create. Each symbol consists of graphical elements (lines, circles, and so forth), SmartText properties, fixed text (labels), connect points, and an origin point.

Note that you can use all the graphical tools provided in Catalog Manager to create your symbols.

Connect Points

Two types of connect points are used in SLDs:

- **Connect Point** - for main power connection.
- **Auxiliary Point** - for creating the logical connection between equipment, metering, and relays.

All symbols must have at least two connect points, apart from some exceptions listed below.

Each connect point consists of a:

- **Connection type** - Connect point or auxiliary point.

- **Connection angle** - The angle at which the connection is made between the equipment and the connector.

- **Side** - A numeric value used to distinguish between the "power in" – 1, and the "power out" – 2, sides of the equipment.

- **Number** - A sequential, unique, index number for the connect point. Each side, within a connection type, starts at number one and progresses in sequential steps.

The upper and lower connect points must have the same X coordinates.

Origin Point

Place the origin point in the middle of the existing or virtual line that connects the upper and lower point.

Special Symbols

The software uses several symbols to represent a transformer in a single line diagram - Transformer symbol and TransformerComponent symbols. You create each symbol separately. However, the origin point of all the symbols has to be placed in such a way that when the software superimpositions all the symbols, it creates a complete representation of a compound transformer.

Use the following properties to create transformer symbols:

- Symbol name - Transformer
- One upper connect point
- Connection angle: 90 degrees (up)
- Index 1
- Symbol name - TransformerComponent0
- One lower connect point
- Connection angle: -90 degrees (down)
- Index 1

- Symbol name - TransformerComponent1

- One lower connect point.

- Connection angle: -90 degrees (down)

- Index 1

- Symbol name - TransformerComponent2

- One lower connect point

- Connection angle: -90 degrees (down)

- Index 1

Catalog Manager uses the TransformerComponent0 symbol together with the Transformer symbol to create a 2-winding transformer representation.

Catalog Manager uses TransformerComponent1 together with the Transformer and TransformerComponent 2 symbols to create a 3-winding transformer representation.

For an example of origin placement in each of the transformer symbols, you can open any of the above mentioned symbol files located in the folder that you define in the Default SLD Symbol Path location in Options Manager.

- Bus symbols:

- Bus - SmartPlant Electrical creates bus representations in SLD automatically. However, you still need to create a bus symbol and set its line type, line width, and define SmartText attributes. The origin point must always be at the left edge of the symbol.

- SmartPlant Electrical uses the above properties to create the bus symbols in single line diagrams. You can create as many bus symbols with different properties as required so that you can generate single line diagrams with various line types and widths. Note that SmartPlant Electrical uses only the symbol properties and not the file itself to represent buses in single line diagrams. After creating bus symbols and defining their properties in Catalog Manager, you can modify the bus properties prior to generating an SLD by associating a different symbol file using the **Associate Custom Symbols** command from the **Actions** menu. This way, the software can generate a single line diagram with various bus representations.

- Symbols that represent the connection between two buses in a PDB:

- Horizontal representation - Special symbols are used to show the horizontal connection between two buses and their internals. The two symbols are **CircuitBusTieCoupler.sym** and **CircuitBusTieRiser.sym**. All disconnect equipment, cable, or any other item that needs to be displayed in the internals of the bus tie have a specific syntax, **<ItemType>BusTie.sym**. The software generates the SLD and searches for the BusTie.sym suffixed symbols if in the **Single Line Diagram Options** dialog box the circuit internals have been chosen to be displayed. Note that the BusTie symbol is a `circuit' type symbol and has its first connection point on the rightmost side and the second connection point at the leftmost side.

- Vertical representation - The standard bus coupler-riser circuit connection representation that includes both coupler and the riser circuits and displays them as vertical objects, connected to each other. The software displays both circuits as standard arrows.

- Other symbols:

- Other special symbols include <**ItemType**>**Alt.sym** used to represent equipment that has two sources of power supply**, Buswrappingmarker_Right**, **Buswrappingmarker_Left**, and **Triangle-Compare** .
  Note that the connect point numbering sequence for an **Alt.sym** symbol is "main power in" – 1, "power out" – 2, and "alternate power in" – 3.

- From - This is a system symbol; do not modify it.

- Default (Symbol name = Default) - SmartPlant Electrical uses a default symbol whenever a required symbol for a particular item type does not exist. Default symbols require two connect points. You can select any item type for the Default symbol.

Notes

- For information on how to use custom symbols in your SLDs and the SLD legends, see *Use Custom Symbols in SLDs*, page 309.

- For information on how to place connect points in a symbol in Catalog Manager, see Place Symbol Connect Points in the *Catalog Manager User's Guide*, under *Working with Symbols*, *Creating Symbols*, *Place Point Command*.

- A list of all shipped symbols and their SmartText attributes is available in the symbols.txt file located in the folder that you define in the Default SLD Symbol Path location in Options Manager.

- SmartPlant Electrical shipped data includes two sets of transformer symbol files. The IEC set includes:

- TransformerIEC

- TransformerComponent0IEC

- TransformerComponent1IEC

- TransformerComponent2IEC

- The NEC set includes:

- TransformerNEC

- TransformerComponent0NEC

- TransformerComponent1NEC

- TransformerComponent2NEC

- Since the software uses the Transformer, TransformerComponent0, TransformerComponent1, and TransformerComponent2 symbols by default, you have to replace this set with the NEC or IEC symbols and rename them without their IEC or NEC suffixes if you want to use symbols belonging to another standard.

- The following limitations of SLD symbol creation exist in Catalog Manager:

- On the **Smart Text Editor** dialog box, you cannot use the **Visible**, **Short Text**, and **Format** features.

- The angles for the symbol connect points are fixed at 90 degrees for the top connect point and -90 degrees for the bottom connect point.

- There is no support of symbol properties in the **Properties** window except for the selection of item types.

- The software does not retain the assigned smart text labels if you change the item type of a symbol for which smart text labels have been created.

Related Topics

Place Symbol Connect Points

1. In Catalog Manager, open the desired .sym file.

2. On the **Catalog Tools** toolbar, click **Place Points** .

3. From the **Connection Type** list, select the connection type required.

4. Click the symbol at the location where you want to add the connect point.

   💡 **Tip**

   • The software displays a dynamic dashed line representing the connection of the new connect point.

5. Orient the dashed line to represent the appropriate connection angle for the new connect point.

6. After the connection angle is correct, click again to place the connect point.

   ✍ **Notes**

   • You can also type the exact angle value in the **Connect angle** field to define the connection angle.

   • You can right-click during this procedure to reset the command.

Cable Block Diagrams: An Overview

Cable Block Diagrams (CBDs) provide a graphical representation of the interconnection of all the control, power, instrumentation, and grounding cables, and their related equipment such as power distribution boards (PDB's), generators, motors, and so forth, within your plant. They show the project cable flow between the various cabinets and equipment of your project.

Cable Block Diagrams are used in the preliminary planning stages of the project life cycle to verify the destination panels for cables as no wiring connections or terminations are needed at this early stage of your project. A CBD is also used for ordering made to order (MTO) cables at the feed stage of your project, and later after completion of your project as part of the maintenance program to show the location of all major cables, panels, junction boxes, and so forth. Use the same equipment and cables in more than one drawing.

SmartPlant Electrical allows you to drag and drop items from the Electrical Index or Electrical Engineer to your drawing, and position them exactly where you want. The software creates a representation of existing cables on the fly, or you can manually connect cables between item connection points. SmartPlant Electrical checks any connection to make sure that it is legal and informs you if there is a problem.

Related Topics
- *Cable Block Diagram Common Tasks*, page 322

Cable Block Diagram Common Tasks

The following tasks are used frequently when working with cable block diagrams.

Create a New Cable Block Diagram

This procedure shows you how to create a new cable block diagram, add items, show already connected cables, and connect cables between the items. Using symbols supplied with the software or custom symbols that you create, you drag and drop items where you want to position them within the drawing. When you connect cables between the different items of equipment, the software checks that the connection point is valid, and makes the connection. If no connection point is available the software does not allow the cable to be connected. For more information, see *Create a New Cable Block Diagram*, page 323.

Open an Existing Cable Block Diagram

This procedure shows you how to open an existing cable block drawing. For more information, see *Open an Existing Cable Block Diagram*, page 324.

Hide Cable Categories

This procedure shows you how to hide (filter) the four categories of cables when working on a drawing. CBDs can become very crowded, so by applying a filter you can hide those cables that you are not working with at that moment, allowing you to concentrate on the job at hand. You can use the filter to show only one category of cable in the working and generated drawing, for example power cables, or to show a combination of cables, such as control and instrumentation cables. The filter can be changed on the fly to reveal other cable category or to hide a particular cable category. For more information, see *Hide Cable Categories*, page 326.

Add Cables and Items to an Existing Cable Block Diagram

This procedure shows you how to add cables and items to an existing cable block diagram. For more information, see *Add Cables and Objects to an Existing Cable Block Diagram*, page 326.

Find a Drawing Item in the Electrical Index or Electrical Engineer

This procedure shows you how to find items that appear on your CBD in the **Electrical Index** or **Electrical Engineer**. For more information, see *Find a Drawing Item in the Electrical Index or Electrical Engineer*, page 326.

Show Related Items

This procedure allows you to show those items that are related to the main items that you want to use in your CBD. Using **Show Related Items** opens a new window that shows just those items that are related to the main item, allowing for quick selection when adding to your CBD. For more information, see *Show Related Items*, page 327.

Disconnect Cables from Items in Cable Block Diagrams

This procedure shows you how to disconnect cables from items on your drawing. You can disconnect a cable from Side 1, Side 2, or both sides. For more information, see *Disconnect Cables from Items in Cable Block Diagrams*, page 328.

Remove Items from a Cable Block Diagrams and Wiring Drawings

This procedure shows you how remove a drawing item, such as a PDB, from your drawing. For more information, see *Remove Items from a Cable Block Diagrams and Wiring Drawings*, page 329.

Related Topic

- *Cable Block Diagrams: An Overview*, page 321

Create a New Cable Block Diagram

1. In the **Electrical Index** open the **Documents** folder.

2. Right-click the **Cable Block Diagrams** folder, and on the shortcut menu, click **New Cable Block Diagram**.

3. From the **Electrical Index** or **Electrical Engineer** drag the items you want on to the cable block diagram (cabinets, PDB's, main transformers, and so forth.), and position them on the drawing.

4. Select any item on the drawing and do one of the following:

   - Click **Activate Connection Mode** .

   - Click **Edit** > **Activate Connection Mode.**

5. On the **Electrical Index** or **Electrical Engineer** select the cable you want to connect between two items.

6. Move the cursor over the drawing to view the available, valid connection points of the items and cable category on your drawing.

   💡 **Tip**

   - The software displays only the connection points available for the specific category of the cable chosen. For more details, see *Place Point Ribbon (SmartPlant Electrical)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command.*

7. Click the connection point of the item you want the cable to connect 'From'.

8. Click the connection point of the item you want the cable to connect 'To'.

9. Select another cable or do the following to view the drawing with the attached cable;

- Click **View** > **Refresh**

- Click ⟳

📝 **Notes**

- CBDs do not show internal items of a PDB, such as circuits. Since cables connect to circuits and not to PDBs, when connecting cables to PDBs in a cable block diagram, you have to associate the cables with the specific circuit using either the cable **Common Properties** or the **Electrical Engineer**.

- **Refresh** also saves the drawing. If you do not use **Refresh** or you make changes to your drawing after refreshing, the software prompts you to save the drawing on closing.

- You can move and reposition any item or cable by selecting and dragging the item to its new position.

- To exit the **Connection Mode** select another command or press the Escape key.

Related Topics
- *Cable Block Diagram Common Tasks*, page 322
- *Cable Block Diagrams: An Overview*, page 321

Open an Existing Cable Block Diagram

1. In the **Electrical Index, Documents Folder** open the **Cable Block Diagrams** folder.

2. Select the Cable Block Diagram that you want to open, and do one of the following.

- Click **File** > **Open** > **Drawing**.

- Click 📂.

- Right-click and on the shortcut menu, click **Open**.

Related Topics
- *Cable Block Diagram Common Tasks*, page 322
- *Cable Block Diagrams: An Overview*, page 321

Hide Cable Types

1. In the **Cable Block Diagrams** folder, select the drawing you want to filter.

2. Do one of the following:

    *   Click **Edit** > **Document Properties**.

    *   Click **Document Properties** .

    *   Right-click the document, and on the shortcut menu, click **Document Properties**.

3. Under **Show cables in drawing** clear the check boxes beside those cables that you do not want to see.

    💡 **Tip**

    *   You can change the filter at any time by going to the **Document Properties** dialog box and selecting or clearing the relevant check boxes.

    Related Topics
    *   *Cable Block Diagram Common Tasks*, page 322
    *   *Cable Block Diagrams: An Overview*, page 321

Add Cables and Items to an Existing Cable Block Diagram

1. Open the cable block diagram that you want to work on.

2. From the **Electrical Index** or **Electrical Engineer** drag the items you want to add to the cable block diagram and position them on the drawing.

3. Select any item on the drawing and do one of the following:

    *   Click **Activate Connection Mode** .

    *   Click **Edit** > **Activate Connection Mode.**

4. On the **Electrical Index** or **Electrical Engineer** select the cable you want to connect between added items.

5. Move the cursor over the drawing to view the available, valid connect points of the items and cable category on your drawing. For more details, see *Place Point Ribbon (SmartPlant Electrical)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command*.

6. Click the connect point of the item you want the cable to connect 'From'.

7. Click the connect point of the item you want the cable to connect 'To'.

8. Select another cable or do the following to view the drawing with the attached cable;

- Click **View** > **Refresh**
- Click

Note

- **Refresh** also saves the drawing. If you do not use **Refresh** or you make changes to your drawing after refreshing, the software prompts you to save the drawing on closing.

- You can move and reposition any item or cable by selecting and dragging the item to its new position.

- To exit the **Connection Mode** select another command or press Esc.

Find a Drawing Item in the Electrical Index or Electrical Engineer

1. In a cable block diagram or wiring drawing, select the item you want to find in the **Electrical Index** or **Electrical Engineer**.

   Tip

   - Make sure that the **Electrical Index** and **Electrical Engineer** are open.

2. Right-click the selected item, and from the shortcut menu select **Find in Electrical Index** or **Find in Electrical Engineer**. The software opens the folder where the item is stored and highlights the item.

Related Topics
- *Cable Block Diagram Common Tasks*, page 322
- *Cable Block Diagrams: An Overview*, page 321
- *Wiring Project Items Common Tasks*, page 336
- *Working with Wiring: An Overview*, page 329

Show Related Items

1. On the **Electrical Index** or **Electrical Engineer** navigate to the item that you want to see its related items.

2. Right-click on the item and from the shortcut menu, select **Show Related Items in a New Window**.

📝 **Notes**

- A new window opens showing the open folders of the selected item and all the related items, you can then drag and drop these items onto your CBD in the same way as from the **Electrical Index** or **Electrical Engineer.**

- To close the **Related Items** window, double-click on the blue title bar at the top of the window and then click on the close icon.

Related Topics
- *Cable Block Diagram Common Tasks*, page 322
- *Cable Block Diagrams: An Overview*, page 321

Disconnect Cables from Items in Cable Block Diagrams

1. Select the cable you want to disconnect.

2. Do one of the following:

   - Click ⮡.

   - Click **Edit** > **Disconnect**.

   - Right-click, and on the shortcut menu, click **Disconnect Items**.

3. At the prompt, select the required check box, and click **OK**.

✎ **Notes**

   - If you disconnect a cable that is connected only on one of its ends, the software disconnects the cable without showing you a prompt to confirm the disconnection.

   - Disconnecting both ends of a cable disconnects the items that it was connecting and removes the cable from the drawing. However, the software does not delete the cable from the project. The software only disconnects the items from each other and retains the association between the item and the cable.

Related Topics
   - *Cable Block Diagram Common Tasks*, page 322
   - *Cable Block Diagrams: An Overview*, page 321
   - *Wiring Project Items Common Tasks*, page 336
   - *Working with Wiring: An Overview*, page 329

Remove Items from a Cable Block Diagrams and Wiring Drawings

1. On the drawing, select the item that you want to remove.

2. Do one of the following:

   - Click ✎.

   - Click **Edit Remove**.

   - Press the **Delete** key.

Related Topics
   - *Cable Block Diagram Common Tasks*, page 322
   - *Cable Block Diagrams: An Overview*, page 321
   - *Wiring Project Items Common Tasks*, page 336
   - *Working with Wiring: An Overview*, page 329

Working with Wiring: An Overview

SmartPlant Electrical enables you to perform wiring and create wiring drawings. The software provides for the creation of basic wiring and enables you to make the terminations that you require. You can do the following:

- Create terminal strips for all types of applicable equipment.

- Create terminals on terminal strips.

- Manage both cable sides.

- Manage both conductor sides.

- Create wiring drawings and make connections.

- Generate termination reports.

SmartPlant Electrical deals with equipment interconnections that are external to PDBs or main equipment item types. The software handles the internal circuit wiring in typical schematics. Cable terminations can reach and include equipment terminal strips.

After creating your wiring items and making the required terminations, you can generate cable wiring schedule and terminal strip connection reports. For details, see *Shipped Reports*, page 390.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Wiring Project Items Common Tasks*, page 336

Creating and Defining Wiring Items Common Tasks

The following tasks are used frequently when you create and define wiring items in SmartPlant Electrical.

Create a Terminal Strip in the Electrical Index

This procedure explains how to create a single terminal strip in the **Electrical Index**. For more information, see *Create a Terminal Strip in the Electrical Index*, page 331.

Create a Terminal in the Electrical Index

This procedure explains how to add a terminal to terminal strip in the **Electrical Index**. For more information, see *Create a Terminal in the Electrical Index*, page 332.

Define a Reference Terminal Strip Configuration

This procedure explains how to define a reference terminal strip configuration that you can apply to a terminal strip. You can create as many reference configurations as you require and store them in the **Reference Data Explorer**. For more information, see *Define a Reference Terminal Strip Configuration*, page 333.

Define a Terminal Strip Configuration for a Single Terminal Strip

This procedure explains how to define a terminal strip configuration for a terminal strip you are currently editing. Note that the software does not retain the configuration that you define. The software uses this configuration to create the terminals on the current terminal strip but does not store the configuration in the database. For more information, see *Define a Terminal Strip Configuration for a Single Terminal Strip*, page 335.

Apply a Reference Terminal Strip Configuration to a Terminal Strip

This procedure explains how to apply a reference terminal strip configuration to a terminal strip. Applying a reference configuration to a terminal strip saves time when creating terminal strips. For more information, see *Apply a Reference Terminal Strip Configuration to a Terminal Strip*, page 334.

Show Related Items in a New Window

SmartPlant Electrical allows you to select an item in the **Electrical Index** or **Electrical Engineer** and show all the items that are related to the selected item in a new window. For example, if you select a motor, the software displays the associated control station and feeder circuits grouped together in a new window. This way you do not need to search for associated items and save time when wiring your project items. For more information, see *Show Related Items in a New Window*, page 43.

Create a Terminal Strip in the Electrical Index

1. In the **Electrical Index**, expand the hierarchy of the **Electrical Equipment** folder and then an individual equipment item to display the **Terminal Strips** folder.

2. Right-click the **Terminal Strips** folder and then click **New Terminal Strip**.

3. If the **Terminal Strip Properties** dialog box does not open automatically, right-click the new terminal strip and then on the shortcut menu, click **Common Properties.**

   💡 **Tip**

   - SmartPlant Electrical opens the **Common Properties** dialog box automatically after creating a new item if you selected the **Common Properties** dialog box option on the **General** tab of the **Preferences** dialog box.

4. On the **Terminal Strip Properties** dialog box, under **Item tag**, type a new name as you require.

5. Type a short description if you require.

6. In the **Terminal strip configuration** group box, do one of the following:

   - Click **Find** to find an existing reference configuration that you can apply to the current terminal strip. For details, see *Apply a Reference Terminal Strip Configuration to a Terminal Strip*, page 334.

   - Define a new terminal strip configuration. For details, see *Define a Terminal Strip Configuration for a Single Terminal Strip*, page 335.

7. Click **OK**.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Working with Wiring: An Overview*, page 329

Create a Terminal in the Electrical Index

1. In the **Electrical Index**, expand the hierarchy of the **Electrical Equipment** folder and then an individual equipment item and its terminal strips.

2. Right-click a terminal strip and then click **New Terminal**.

3. If the **Terminal Properties** dialog box does not open automatically, right-click the new terminal and then on the shortcut menu, click **Common Properties.**

   💡 **Tip**

   - SmartPlant Electrical opens the **Common Properties** dialog box automatically after creating a new item if you selected the **Common Properties** dialog box option on the **General** tab of the **Preferences** dialog box.

4. On the **General** tab of the **Terminal Common Properties** dialog box, under **Item tag**, type a name for the new terminal as you require.

5. Type a short description if you require.

6. Select a terminal type that you require. You can define and customize types in Data Dictionary Manager.

7. Select a terminal color that you require. You can define and customize terminal colors in Data Dictionary Manager.

8. Under **Sequence**, type a value that sets the sequential number of the current terminal on its terminal strip.

9. Click the **Procurement** tab.

10. Select a manufacturer and a model as you require. You can define and customize these entries in Data Dictionary Manager.

11. Type a store part number if you require.

Define a Reference Terminal Strip Configuration

1. In the **Reference Data Explorer**, right-click the **Terminal Strip Configurations** folder.

2. On the shortcut menu, click **New Terminal Strip Configuration**.

3. On the **Terminal Strip Configuration** dialog box, type the configuration name and description as you require.

4. In the **Configuration properties** group box, enter the total number of terminals.

5. Under **First terminal**, enter an appropriate value.

6. In the **Terminal strip pattern** data grid, click and type in the **Prefix** field, click and type a value if you want to prefix the names of the terminals.

7. Select the **Numbered** check box if you want the number of the terminal to appear between the prefix and the suffix.

8. Select the **Incremented** check box if you want the terminal number to be incremented within the pattern.

9. In the **Suffix** field, click and type a value if you want a suffix for the names of the terminals.

10. Click **Add** if you want to add another data row in the terminal strip pattern.

    💡 **Tip**

    - The maximum number of rows that you can add cannot exceed the total number of terminals that you enter.

11. Click **OK**.

Related Topic
- *Creating and Defining Wiring Items Common Tasks*, page 330

Apply a Reference Terminal Strip Configuration to a Terminal Strip

1. In the **Electrical Index** or **Reference Data Explorer**, expand the hierarchy of the **Electrical Equipment** folder as necessary to display the **Terminal Strips** folder.

2. Expand the **Terminal Strips** folder and then right-click a terminal strip that you require.

3. On the shortcut menu click **Common Properties**.

4. On the **Terminal Strip Properties** dialog box, click **Configuration**.

5. On the **Terminal Strip Configuration** dialog box, click **Find**.

6. On the **Find** dialog box, click **Find Now**.

7. In the **Results** data window, select an appropriate terminal strip configuration and click **OK**.

8. On the **Terminal Strip Configuration** dialog box, click **OK**.

Notes

- The software does not allow you to apply a reference terminal strip configuration to a terminal strip that has already been wired.

- If you apply a reference terminal strip configuration to a terminal strip that already has a terminal strip configuration, the replaces the existing configuration with the reference configuration that you are applying.

Related Topics

Define a Terminal Strip Configuration for a Single Terminal Strip

1. On the **Terminal Strip Properties** dialog box, under **Terminal strip configuration**, enter the total number of terminals.

2. Under **First terminal**, enter an appropriate value.

3. In the **Terminal strip pattern** data grid, click and type in the **Prefix** field, click and type a value if you want to prefix the names of the terminals.

4. Select the **Numbered** check box if you want the number of the terminal to appear between the prefix and the suffix.

5. Select the **Incremented** check box if you want the terminal number to be incremented within the pattern.

6. In the **Suffix** field, click and type a value if you want a suffix for the names of the terminals.

7. Click **Add** if you want to add another data row in the terminal strip pattern.

    💡 **Tip**

    - The maximum number of rows that you can add cannot exceed the total number of terminals that you enter.

8. Click **OK**.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Working with Wiring: An Overview*, page 329

Wiring Project Items Common Tasks

SmartPlant Electrical enables you to effect wiring termination of project items. You can create wiring diagrams where you make the terminations that you require. Also, can generate various wiring reports that show the wiring of the project items.

Note that you can only wire and terminate those cables that have already been associated with the electrical items that you want to wire. Therefore, before terminating the cable wires, make sure that you associate the required cables with appropriate project items.

The software provides you with several ways to wire your project items. You can choose one of the following termination methods:

- Continuous connection — Allows you to perform wiring in a sequential order starting with a terminal that you select.

- Custom connection — Allows you to select a reference custom connection type for the terminals that you want to wire.

- Batch cable connection — select a number of cables that have the same wiring pattern. You can connect these cables in batch mode to project items that have the same connectivity.

Note that when you change the association a terminated power cable by dragging it to another equipment item in the **Electrical Engineer**, disconnects this cable from the terminal strip of that equipment. Changing the association of a terminated non-power cable on the **Cable Common Properties** dialog box also results in disconnection. The software disconnects the cable without displaying a warning message.

The following tasks are used frequently when you wire project items in SmartPlant Electrical.

Define a Connection Type

This option allows you to select, create, or edit a connection type in the **Reference Data Explorer**. A connection type facilitates rapid cable set connections. You define a connection type that contains a connection pattern. A connection type contains the definition of the pattern by which wires will be connected on a terminal strip, and allows automatic connection of successive cable sets. Once a wiring pattern has been defined, you can select it when you wire project items using the custom connection method. The connection type that you select during the connection procedure will determine how the connections are made. For more information, see *Define a Connection Type*, page 337.

Wire Project Items Using the Continuous Connection Method

This procedure explains how to make connections in a wiring drawing using the continuous connection method. The continuous connection method allows you to make a connection in a sequential way by selecting a terminal that will be wired first. The software then automatically wires the rest of the terminals in a sequential order. For more information, see *Wire Project Items Using the Continuous Connection Method*, page 340.

Wire Project Items Using the Custom Connection Method

This procedure explains how to make connections in a wiring drawing using the custom connection method. The custom connection method allows you to make your connections based on an existing reference connection type of a selected cable. You start by selecting a terminal to be wired first and the software then connects the rest of the terminals according to the connection type. For more information, see *Wire Project Items Using the Custom Connection Method*, page 339.

Connect Project Cables in Batch Mode

This feature enables you to connect in batch mode numerous project cables that belong to the same category, for example, power, control, instrumentation, and so forth. For example, you can you can terminate all the project control cables that are associated with 10hp motors and their circuits. The software allows you to select a connection method (continuous or custom) for the batch operation. For more information, see *Connect Project Cables in Batch Mode*, page 343.

Connect a Single Conductor

This procedure shows you how to connect a single conductor. For more information, see *Connect a Single Conductor*, page 346.

Connect a Cable Set

This procedure shows you how to connect a single cable set. For more information, see *Connect a Cable Set*, page 345.

Open an Existing Wiring Drawing

This procedure shows you how to open an existing wiring drawing. For more information, see *Open an Existing Wiring Drawing*, page 348.

Find a Drawing Item in the Electrical Index or Electrical Engineer

This procedure shows you how to find items that appear in your wiring diagram in the **Electrical Index** or **Electrical Engineer**. For more information, see *Find a Drawing Item in the Electrical Index or Electrical Engineer*, page 326.

Disconnect a Cable, Cable Set, or Conductor in a Wiring Drawing

This procedure shows you how to disconnect cables, cable sets, or conductors in a wiring drawing. When disconnecting a cable, the software disconnects it on both sides and removes it from the drawing. However, the cable still remains associated with the wiring items even if it has been disconnected. When disconnecting a cable set or a conductor, the software also removes the disconnected set or conductor from the drawing. For more information, see *Disconnect a Cable, Cable Set, or Conductor in a Wiring Drawing*, page 347.

Remove Items from a Cable Block Diagrams and Wiring Drawings

This procedure shows you how remove an item, such as a PDB, from your drawing. For more information, see *Remove Items from a Cable Block Diagrams and Wiring Drawings*, page 329.

Related Topics
- *Working with Wiring: An Overview*, page 329

Define a Connection Type

1. In the **Reference Data Explorer**, right-click the **Connection Types** folder.

2. On the shortcut menu, click **New Connection Type**.

3. On the **Connection Type** dialog box, type a connection type name and description.

4. For instrumentation cables, in the **Skip between sets** field, enter the number of terminals that will be skipped between the first terminals of each cable set. These terminals will be left unconnected.

5. In the **Conductors** group box, do the following:

   a. Click in the **Skip** text field and type the number of terminals to skip before connecting this conductor in the cable set. The skip refers to the number of terminals that will be left unconnected between consecutive cable sets. Enter the number of terminals to be skipped according to your needs. In most cases, this value is incremented for successive conductors. You can accept the displayed value if suitable.

   b. In the **Land Side** field, select the **A – Active** or **O – Opposite** check box to select the terminal side to which you want to connect each conductor. The side that is currently active depends on the side that you start connecting.

   💡 **Tip**

   - In the environment of a wiring drawing, the **A** side is the active side of the terminal that you want to connect. It can be its left or right side, depending on which terminal side you clicked after you moved the cursor over the connection points. The **O** side is the opposite side of the active side.

6. Click **Add** to add another row to the **Conductors** data window.

   💡 **Tips**

   - The **Conductor** field signifies a specific conductor in a cable set. The displayed **Wire00x** values represent the conductor names and are used to illustrate the current connection type configuration. This field is not editable.

   - The number of rows that you add has to match the number of conductors that will be connected. The software does not connect the leftover conductors that do not have connection type definitions in the **Conductors** data window.

7. Click **OK**.

Related Topics
   - *Creating and Defining Wiring Items Common Tasks*, page 330
   - *Wiring Project Items Common Tasks*, page 336
   - *Working with Wiring: An Overview*, page 329

Wire Project Items Using the Custom Connection Method

1. In the **Electrical Index**, expand the **Documents** folder.

2. Right-click the **Wiring Drawings** folder and then on the shortcut menu, click **New Wiring Drawing**.

3. Drag the project items that you want to wire from any open window to the new wiring drawing.

   💡 **Tips**

   - You can drag any appropriate item such as a motor, circuit, panel, and so forth.

   - You can drag your items from the **Electrical Index**, **Electrical Engineer**, or **Related Items** window.

4. Do one of the following:

   - Click **Activate Connection Mode** 🔧.

   - Click **Edit** > **Activate Connection Mode.**

5. In the **Electrical Index**, **Electrical Engineer**, or **Related Items** window, select an appropriate cable that you want to terminate.

   💡 **Tip**

   - You can connect only those cables that are associated with equipment on one or both cable sides.

6. In the wiring drawing, move the cursor over the terminal strip that you want to connect to the cable.

   💡 **Tip**

   • The software displays the available, valid connection points of the terminal strip on your drawing. For more details, see *Place Point Ribbon (SPEL)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command*.

7. Click the connection point of the terminal to which you want to connect the first conductor on the appropriate side.

   💡 **Tips**

   • You can connect the first conductor to any terminal on the current terminal strip. The software will then connect the remaining conductors according to the custom connection type that you select. (See the following steps in this procedure.)

   • The number of conductors that you can connect cannot exceed the number of available wiring connection points on a terminal.

   • You can connect only one conductor on a wiring connection point.

8. On the shortcut menu, click **Custom Connection**.

9. On the **Cable Connection Type Selection Dialog Box** dialog box, select a connection type you require.

10. Click **Connect**.

📝 **Notes**

   • The software provides a set of default symbols for terminal strips, terminals, first terminals, cables, cable sets, and conductors. You can customize your own wiring symbols in Catalog Manager. For details, see Creating Symbols: An Overview in the *Catalog Manager User's Guide*, under *Working with Symbols*, *Creating Symbols*.

   • You can perform various actions in a wiring drawing, such as moving items by dragging them to position the items in the drawing, removing items from the drawing, disconnecting cables, and so forth. For details, see the Table of Contents or *Wiring Project Items Common Tasks*, page 336.

   • Removing a wiring item from a drawing does not delete that item from your project.

   • Disconnecting two wiring items in a drawing, disconnects the items in your project, removes the connecting cable from the drawing, and retains the association between the cable and the items in your project. For details, see *Disconnect Cables from Items in Cable Block Diagrams and Wiring Drawings*, page 329.

- Clicking **Refresh** saves the drawing. If you do not click **Refresh** or you make changes to your drawing after refreshing, the software prompts you to save the drawing on closing.

- To exit the **Connection Mode** select another command or press the Escape key.

- You can add other wiring items to the drawing by dragging them to drawing from the **Electrical Index** or **Related Items** window.

- After adding a wiring item that is connected to a cable, the drawing displays the item connected to the cable on one of the cables sides and the name of the item tag of the wiring item that is connected on the other cable side. After dragging that wiring item to the drawing, software replaces the item tag name with the symbol of that wiring item.

- The software allows you to add annotations and use redlining in your wiring drawings. For details, see *Managing Annotations and Redlining: An Overview*, page 356.

- If you move a macro to another place in the drawing, clicking a macro highlights the cable, cable set, or conductor that is associated with that macro. This way you can see which cable, cable set, or conductor belongs to that macro.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Wiring Project Items Common Tasks*, page 336
- *Working with Wiring: An Overview*, page 329

Wire Project Items Using the Continuous Connection Method

1. In the **Electrical Index**, expand the **Documents** folder.

2. Right-click the **Wiring Drawings** folder and then on the shortcut menu, click **New Wiring Drawing**.

3. Drag the project items that you want to wire from any open window to the new wiring drawing.

   ### 💡 Tips

   - You can drag any appropriate item such as a motor, circuit, panel, and so forth.

   - You can drag your items from the **Electrical Index**, **Electrical Engineer**, or **Related Items** window.

4. Do one of the following:

   - Click **Activate Connection Mode** 🔤.

   - Click **Edit** > **Activate Connection Mode.**

5. In the **Electrical Index**, **Electrical Engineer**, or **Related Items** window, select an appropriate cable that you want to terminate.

   ### 💡 Tip

   - You can connect only those cables that are associated with equipment on one or both cable sides.

6. In the wiring drawing, move the cursor over the terminal strip that you want to connect to the cable.

   ### 💡 Tip

   - The software displays the available, valid connection points of the terminal strip on your drawing. For more details, see *Place Point Ribbon (SPEL)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command*.

7. Click the connection point of the terminal to which you want to connect the first conductor on the appropriate side.

    💡 **Tips**

    - You can connect the first conductor to any terminal on the current terminal strip. The software will then automatically connect the remaining conductors in a sequential order.

    - The number of conductors that you can connect cannot exceed the number of available wiring connection points on a terminal.

    - You can connect only one conductor on a wiring connection point.

8. On the shortcut menu, click **Continuous Connection**.

9. In the wiring drawing, move the cursor over the connection points of the second terminal strip.

10. On the shortcut menu, click **Continuous Connection**.

📝 **Notes**

- The software provides a set of default symbols for terminal strips, terminals, first terminals, cables, cable sets, and conductors. You can customize your own wiring symbols in Catalog Manager. For details, see Creating Symbols: An Overview in the *Catalog Manager User's Guide*, under *Working with Symbols*, *Creating Symbols*.

- You can perform various actions in a wiring drawing, such as moving items by dragging them to position the items in the drawing, removing items from the drawing, disconnecting cables, and so forth. For details, see the Table of Contents or *Wiring Project Items Common Tasks*, page 336.

- Removing a wiring item from a drawing does not delete that item from your project.

- Disconnecting two wiring items in a drawing, disconnects the items in your project, removes the connecting cable from the drawing, and retains the association between the cable and the items in your project. For details, see *Disconnect Cables from Items in Cable Block Diagrams and Wiring Drawings*, page 329.

- Clicking **Refresh** saves the drawing. If you do not click **Refresh** or you make changes to your drawing after refreshing, the software prompts you to save the drawing on closing.

- To exit the **Connection Mode** select another command or press the Escape key.

- You can add other wiring items to the drawing by dragging them to drawing from the **Electrical Index** or **Related Items** window.

- After adding a wiring item that is connected to a cable, the drawing displays the item connected to the cable on one of the cables sides and the name of the item tag of the wiring item that is connected on the other cable side. After dragging that wiring item to the drawing, software replaces the item tag name with the symbol of that wiring item.

- The software allows you to add annotations and use redlining in your wiring drawings. For details, see *Managing Annotations and Redlining: An Overview*, page 356.

- If you move a macro to another place in the drawing, clicking a macro highlights the cable, cable set, or conductor that is associated with that macro. This way you can see which cable, cable set, or conductor belongs to that macro.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Wiring Project Items Common Tasks*, page 336
- *Working with Wiring: An Overview*, page 329

Connect Project Cables in Batch Mode

1. In the **Electrical Index**, click the **Cables** folder.

2. In the **Item Tag** pane, select the cables you want to terminate in batch mode.

   Tip

   - You can only connect cables in batch mode if they belong to the same category, for example, power, control, instrumentation, and so forth.

3. Click **Actions** > **Cables** > **Batch Cable Connection**.

4. On the **Batch Cable Connection** dialog box, select the **Activate cable side 1 (From)** check box, **Activate cable side 2 (To)** check box, or both check boxes.

5. Under **Terminal strip sequence**, type the sequence of the appropriate terminal strip that is associated with the equipment you want to connect. The value you type defines the terminal strip to be connected. Type a value for one or both cable sides as you require.

6. Under **First terminal**, type a value to instruct the software which terminal on the current terminal strip is to be connected first.

7. Select an appropriate active connection side for one or both cable ends.

8. From the **Connection type** list for one or both cable sides, select an appropriate connection type.

    💡 **Tip**

    -   From the **Connection type** list, you can select **Continuous** (wire the first selected terminal of an item and then automatically wire the rest of the terminals) or a name of an existing custom connection type. If the connection type that you require is not on the list, create it in the **Reference Data Explorer**. For details, see *Define a Connection Type*, page 337.

9. In the **Cable connection details** data window, select the **Select** check box next to the appropriate cables that you want to connect.

10. Click **OK**.

Related Topics
-   *Creating and Defining Wiring Items Common Tasks*, page 330
-   *Wiring Project Items Common Tasks*, page 336
-   *Working with Wiring: An Overview*, page 329

Connect a Cable Set

1. Do one of the following:

    -   Click **Activate Connection Mode** 🔽.

    -   Click **Edit** > **Activate Connection Mode.**

2. In **Electrical Index**, **Electrical Engineer**, or **Related Items** window, expand the hierarchy of a cable to display its sets.

3. Select a cable set.

4. In the wiring drawing, move the cursor over the terminal strip that you want to connect to the cable set.

    💡 **Tip**

    -   The software displays the available, valid connection points of the terminal strip on your drawing. For more details, see *Place Point Ribbon (SPEL)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command*.

5. Click the connection point of the terminal to which you want to connect the cable set.

6. On the shortcut menu, click **Custom Connection**.

   💡 **Tip**

   • If you select the **Continuous Connection Method**, the software connects the cable set starting with the terminal you clicked and completes the connection of the other terminals in a sequential order.

7. On the **Cable Connection Type Selection Dialog Box** dialog box, select a connection type you require.

8. Click **Connect**.

Related Topics
   • *Creating and Defining Wiring Items Common Tasks*, page 330
   • *Wiring Project Items Common Tasks*, page 336
   • *Working with Wiring: An Overview*, page 329

Connect a Single Conductor

1. Do one of the following:

   • Click **Activate Connection Mode** 🔧.

   • Click **Edit** > **Activate Connection Mode.**

2. In **Electrical Index**, **Electrical Engineer**, or **Related Items** window, expand the hierarchy of a cable to display its conductors.

3. Select a conductor.

4. In the wiring drawing, move the cursor over the terminal strip that you want to connect to the conductor.

   💡 **Tip**

   • The software displays the available, valid connection points of the terminal strip on your drawing. For more details, see *Place Point Ribbon (SPEL)* in the *Catalog Manager User's Guide*, under *Working with Symbols > Place Point Command*.

5. Click the connection point of the terminal to which you want to connect the conductor.

   📝 **Note**

   • In a wiring drawing, the software does not show the item tag label for conductors whose polarity property is defined as shield.

Related Topics
   • *Creating and Defining Wiring Items Common Tasks*, page 330
   • *Wiring Project Items Common Tasks*, page 336
   • *Working with Wiring: An Overview*, page 329

Disconnect a Cable, Cable Set, or Conductor in a Wiring Drawing

1. In a wiring drawing, select a cable, cable set, or conductor that you want to disconnect.

2. Do one of the following:

   - Click ⬚.

   - Click **Edit** > **Disconnect**.

   - Right-click, and on the shortcut menu, click **Disconnect Items**.

📝 **Notes**

   - After you confirm the disconnection of a cable, the software disconnects the cable on both sides and removes it from the drawing. However, the software does not delete the cable from the project and retains the association between the wiring items and the cable.

   - When disconnecting cable sets or conductors, the software removes these items from the drawing.

Related Topics
   - *Wiring Project Items Common Tasks*, page 336
   - *Working with Wiring: An Overview*, page 329

Open an Existing Wiring Drawing

1. In the **Electrical Index**, click **Documents** > **Wiring Drawings**.

2. Select a wiring drawing and do one of the following:

   - Click **File** > **Open** > **Drawing**.

   - Click 📂.

   - Right-click and on the shortcut menu, click **Open**.

📝 **Notes**

   - When opening a wiring drawing that contained a wiring item that has been deleted from your project, the wiring drawing no longer displays this item, showing the cable and wiring item connected to the other cable end.

   - If you deleted both wiring items that were connected to a cable, the wiring drawing displays the cable only.

   - If you deleted a cable that was used to connect two wiring items, the wiring diagram shows the two items as disconnected.

- Removing a wiring item from a drawing does not delete that item from your project.

- Disconnecting two wiring items in a drawing, disconnects the items in your project, removes the connecting cable from the drawing, and retains the association between the cable and the items in your project.

- You can add other wiring items to the drawing by dragging them to drawing from the **Electrical Index** or **Related Items** window.

- After adding a wiring item that is connected to a cable, the drawing displays the item connected to the cable on one of the cables sides and the name of the item tag of the wiring item that is connected on the other cable side. After dragging that wiring item to the drawing, software replaces the item tag name with the symbol of that wiring item.

- The software allows you to add annotations and use redlining in your wiring drawings. For details, see *Managing Annotations and Redlining: An Overview*, page 356.

- If you move a macro to another place in the drawing, clicking a macro highlights the cable, cable set, or conductor that is associated with that macro. This way you can see which cable, cable set, or conductor belongs to that macro.

- In a wiring drawing, the software does not show the item tag label for conductors whose polarity property is defined as shield.

Related Topics
- *Creating and Defining Wiring Items Common Tasks*, page 330
- *Wiring Project Items Common Tasks*, page 336
- *Working with Wiring: An Overview*, page 329

Symbology: An Overview

Symbology provides graphical clarity to a drawing by differentiating among various items by their appearance. Symbology refers to the color, line width, shape, and style associated with items.

You can use line widths, styles, shapes, and color to differentiate among the different types of drawing items, such as cables, power distribution boards (PDB), equipment, bus ties and their internals, and so forth. For example, you could define a power cable that connects to a PDB as being a black, 2-mm thick continuous line, where as a control cable could be a blue, 1-mm thick, broken line.

Along with the default symbols supplied with the software, you can create your own customized symbols that can be associated with a document and item tag.

The three different types of documents, SLDs, PDBs, and Wiring, support the use of multiple symbols for the same item by the use of filters, so that a DC motor can be displayed differently than an AC motor in the same drawing.

Using symbology you can customize the style of the PDB frame, add and change the macros, and change their position in relation to the frame.

You must create your custom symbols in the Catalog Manager before using them in SmartPlant Electrical. For more details, see Creating Symbols: An Overview in the *Catalog Manager User's Guide*, under *Catalog Manager*, *Working with Symbols*, *Creating Symbols*.

Related Topics
- *Custom Symbology Common Tasks*, page 350

Custom Symbology Common Tasks

The following tasks are used when associating custom symbols with item tags and documents.

Associate a Custom Symbol with an Item Tag

This procedure shows you how to associate a custom symbol, that has already been created in the Catalog Manager, with an **Item Tag** and **Document Type**. For more information, see *Associate a Custom Symbol with an Item Tag*, page 351.

Dissociate a Symbol from an Item Tag

This procedure shows you how to dissociate a custom symbol from an **Item Tag** and **Document Type**. For more information, see *Dissociate a Symbol from an Item Tag*, page 351.

Related Topics
- *Symbology: An Overview*, page 348

Associate a Custom Symbol with an Item Tag

1. In the **Electrical Index** or **Electrical Engineer**, select the item tag you want to associate with a custom symbol.

2. Do one of the following:

   - Click **Actions > Associate Custom Symbols.**

   - Click .

   - Right-click > **Associate Custom Symbols.**

3. On the **Associate Custom Symbols** dialog box, under **Document Type** click on a cell, and from the list select the document type you require.

4. Under **Symbol**, click .

5. Browse to the symbol (.sym) file you want to add, and select the file.

6. Click **OK**.

✎ **Note**

- You can only choose symbols that are stored in the default folder, and it's sub-folders. You set the symbols default folder in the SmartPlant Electrical Options Manager. For more details, see Defining File Locations: An Overview in the *SmartPlant Electrical Option Manager Users Guide*, under *Using Options Manager*, *Defining File Locations*, *Defining File Locations: An Overview*.

Dissociate a Symbol from an Item Tag

1. In the **Electrical Index** pane, highlight the item tag you want to dissociate the custom symbol from.

2. Do one of the following:

   • Click **Actions > Associate Custom Symbols.**

   • Click .

   • Right-click > **Associate Custom Symbols.**

3. On the **Associate Custom Symbols** dialog box, click on the row you want to remove from the item tag.

4. Click **Remove**.

5. Click **OK**.

Note

   • Removing a symbol disassociates the symbol from the document type and item tag, it does not delete it from the template.

Related Topics

Custom Folders: An Overview

As your database in SmartPlant Electrical expands, the need to organize the various item types into sub-categories becomes apparent. Using custom folders you can create sub-folders for each item type folder. You can use these folders to divide item types into smaller more manageable sections, for example within **Wiring Equipment** > **Cables** > **Power Cables** you could divide the cables by manufacturer, cable specification, or insulation material. Using filters you can place power cables into more than 1 folder, for example you could create a folder for all power cables manufactured by a specific company, and another folder for all power cables that meet NEC specifications, a power cable manufactured by the specific company and that has an NEC specification appears in the folder created for the manufacturer, and appears in the folder created for NEC specification power cables, along with all other power cables that have NEC specifications. The software creates the duplicate items without moving the original item from the Electrical Index or the Reference Data Explorer.

Related Topics
- *Custom Folders Common Tasks*, page 353

Custom Folders Common Tasks

The following tasks are used frequently when creating custom folders in your plant.

Create a Custom Folder

This procedure explains how to create a custom folder, as a sub-folder, in an item type folder. For more information, see *Create a Custom Folder*, page 354.

Apply a Filter to a Custom Folder

This procedure explains how to apply a filter to a custom folder. For more information, see *Apply a Filter to a Custom Folder*, page 354.

Delete a Custom Folder

This procedure explains how to delete a custom folder. For more information, see *Delete a Custom Folder*, page 355.

Related Topics
- *Custom Folders: An Overview*, page 352

Create a Custom Folder

1. In the **Reference Data Explorer** or **Electrical Index,** expand the hierarchy and highlight the item type folder to which you want to add a custom folder.

2. Do one of the following:

   - Click **File** > **New** > **Custom Folder**.

   - Right-click the item type folder, and on the shortcut menu, click **New Custom Folder**.

3. In the **Name** field type the name of the new folder.

4. Next to the **Filter** field, click **Browse** to open the **Select Filter** dialog box, and do one of the following:

   - Select a filter and click **OK**.

   - Create a new filter. For more information on creating a filter, see Using Filters: An Overview in the *Filter Manager User's Guide*, under *Using Filters: An Overview*.

📝 **Notes**

- Items that match the filter criteria are automatically created in the custom folder, without moving them from the original folder.

- Some standard folders already have a hard-coded filter, for example the power cables folder has a hard-coded filter **CableCategory** = *<Property>*. When a hard-coded filter already exists, the filter definition has to be set correctly or this could result in incorrect items appearing in your custom folder. The filter definition for the custom folder should be **Match all**, or the criteria of the existing hard-coded filter should be included in the filter for the custom folder. For example; CableCategory = Power Cable and Conductor Arrangement = 3 core cable.

Apply a Filter to a Custom Folder

1. To apply a filter to a custom folder do one of the following:

   - Create a new custom folder and open the **Select Filter** dialog box. For details, see *Create a Custom Folder*, page 354.

2. Right-click an existing custom folder and on the shortcut menu, click **Properties**. On the **Custom Folder Properties** dialog box, click **Browse**.

3. On the **Select Filter** dialog box, open the folder that contains the required filter.

4. Select the required filter, and click **OK**.

5. On the open dialog box, click **OK**.

📝 **Notes**

- You create filters using Filter Manager, for more information, see Using Filters: An Overview in the *Filter Manager User's Guide*, under *Using Filters: An Overview*.

- Some standard folders already have a hard-coded filter, for example the power cables folder has a hard-coded filter **CableCategory** = *<Property>*. When a hard-coded filter already exists, the filter definition has to be set correctly or this could result in incorrect items appearing in your custom folder. The filter definition for the custom folder should be **Match all**, or the criteria of the existing hard-coded filter should be included in the filter for the custom folder. For example; CableCategory = Power Cable and Conductor Arrangement = 3 core cable.

Related Topics

Delete a Custom Folder

1. In the **Reference Data Explorer** or **Electrical Index** expand the hierarchy and find the custom folder you want to delete.

2. Right-click the folder you want to delete and on the shortcut menu, click **Delete Custom Folder**.

3. At the prompt, click **Yes**.

Note

- Deleting a custom folder does not remove the custom folders contents from the database. These items can still be found in the items parent folder.

Related Topics

Managing Annotations and Redlining: An Overview

You can mark up single line diagrams, schematic drawings, cable block diagrams, and wiring drawings using text or symbols as annotations. The software saves annotations in the same layer as the graphical elements of the drawing

You can lock an annotation to a selected item tag so if you move the item tag, the software also moves the annotation, or you can set the annotation to be moved independently of the item tag.

When working with annotations, you can do the following:

- *Insert Annotation Text*, page 357
- *Insert Annotation Symbols*, page 358
- *Edit Annotation Text*, page 359
- *Delete Annotations*, page 359

Redlining appears in a separate drawing layer and you can hide and display redlining as you require. If a drawing has more than one sheet, you can create redlining and hide or display it independently for each sheet. Therefore, while working in redlining mode, you cannot access any items in the main drawing layer. The software saves the drawing with the redlining items. When opening an SLD or schematic that contains redlining items, the software automatically displays the drawing in redlining mode. You can switch back to regular mode and modify the drawing as needed. The software saves the existing redlining items regardless of whether you open the drawing in redlining mode or not. You can add text, symbols, lines, circles, rectangles, and watermarks as redlining.

Note

- In schematic drawings for which the software also retrieves Enhanced SmartLoop data from SmartPlant Instrumentation, you can add annotations and redlining to the Enhanced SmartLoop drawing sheets.

When working in redlining mode, you can do the following:

- *Insert Redlining Text*, page 360
- *Insert Redlining Graphical Elements*, page 361
- *Edit Redlining Text*, page 362
- *Delete Redlining*, page 362

Related Topics
- *Working with Schematics: An Overview*, page 270
- *Working with Single Line Diagrams Common Tasks*, page 294

Insert Annotation Text

1. Open a single line diagram or a schematic drawing.

2. Do one of the following:

   - Click **Edit > Insert** > **Text**.

   - On the toolbar, click $\boxed{T}$.

3. On the **Text Properties** dialog box, click the **General** tab.

4. Type the annotation text in the **Caption** box.

   💡 **Tip**

   - You can type an unlimited number of lines with up to 28 characters on each line.

5. Select **Set as watermark** if you want to define the text as a watermark that appears in a layer under other drawing items.

6. Under **Annotation locking**, select **Lock to item tag** to lock the position of the annotation text to the selected item tag. In this case, when moving the item tag, the attached annotation moves with it.

7. Select **Move independently** if you want to be able to move the annotation text independently from the item tag that it is locked to. For this option, the software keeps the annotation text locked to the item tag, but allows you to move the annotation text too.

   💡 **Tip**

   - The **Annotation locking** and **Move independently** properties are only available if you first selected an item tag before inserting text.

8. Click the **Properties** tab.

9. In the **Horizontal** and **Vertical** lists, select options to align text within the text box boundaries.

10. Select **Show border** if you want to display the text box border.

11. To rotate the annotation text box, do one of the following in the **Rotation** box:

    - Enter a positive value in degrees to rotate the macro attribute or label clockwise (CW).

    - Enter a negative value in degrees to rotate the macro attribute or label counter-clockwise (CCW).

    💡 **Tip**

    - You can enter values that include decimals, such as 22.5°.

12. From the **Orientation** list, select a text box orientation if needed.

   💡 **Tip**

   - In the vertical orientation, the software rotates the letters 45° CCW.

13. Click **Font** to open a dialog box where you can specify the text font, font style, size, effects, color, and script.

14. Click a blank area on the drawing sheet to position the text.

   💡 **Tip**

   - You can select several text boxes by holding the **Ctrl** key while clicking each text box and then dragging the selected text boxes to the desired location in the drawing.

15. Click **File > Save**.

Insert Annotation Symbols

1. Open a single line diagram or a schematic drawing.

2. Click **Edit > Insert > Symbol** and then navigate to the desired .sym file.

3. Click on the drawing to position the symbol.

4. Click **File > Save**.

Edit Annotation Text

1. Open a single line diagram or a schematic drawing.

2. Select an annotation text box.

3. Do one of the following:

   - Click **Edit** > **Text**.

   - On the toolbar, click ⬚.

4. On the **Text Properties** dialog box, click the **General** tab.

5. Modify the annotation text in the **Caption** box.

   💡 **Tips**

   - You can type an unlimited number of lines with up to 28 characters in each line.

   - When you modify an annotation, the option **Set as watermark** is not available. You can set this option only when defining the text properties for the first time.

6. Click the **Properties** tab.

7. In the **Horizontal** and **Vertical** lists, select options to align text within the text box boundaries.

8. Select **Show border** if you want to display the text box border.

9. To rotate the annotation text box, do one of the following in the **Rotation** box:

   - Enter a positive value in degrees to rotate the macro attribute or label clockwise (CW).

   - Enter a negative value in degrees to rotate the macro attribute or label counter-clockwise (CCW).

   Tip

   - You can enter values that include decimals, such as 22.5°.

10. From the **Orientation** list, select a text box orientation if needed.

    Tip

    - In the vertical orientation, the software rotates the letters 45° CCW.

11. Click **Font** to open a dialog box where you can specify the text font, font style, size, effects, color, and script.

12. Click **File** > **Save**.

Related Topics
   - *Delete Annotations*, page 359
   - *Insert Annotation Symbols*, page 358
   - *Insert Annotation Text*, page 357
   - *Managing Annotations and Redlining: An Overview*, page 356

Delete Annotations

1. Open a single line diagram or a schematic drawing.

2. Select an annotation item on the drawing sheet.

   Tip

   - You can select several items at a time. To do so, hold the **Ctrl** key and click each item. The font and border color of the annotations change when you select them.

3. Press the **Delete** key.

4. Click **File** > **Save**.

Note

   - When you delete an annotation, it is not pasted to the Clipboard and you cannot reinsert it. You cannot restore a deleted annotation after you have saved the file.

Insert Redlining Text

1. Open a single line diagram or a schematic drawing.

2. To activate redlining mode, click **Edit** > **Redlining**.

   Tip

   • When redlining mode is switched on, a check mark appears beside the menu command, and also, the text 'Redlining' appears as the label for the selected drawing sheet.

3. Do one of the following:

   • Click **Edit** > **Insert > Text**.

   • On the toolbar, click [T].

4. On the **Text Properties** dialog box, click the **General** tab.

5. Type the annotation text in the **Caption** box.

   Tip

   • You can type an unlimited number of lines with up to 28 characters on each line.

6. Select **Set as watermark** if you want to define the text as a watermark that appears in a layer under other drawing items.

7. Click the **Properties** tab.

8. In the **Horizontal** and **Vertical** lists, select options to align text within the text box boundaries.

9. Select **Show border** if you want to display the text box border.

10. To rotate the annotation text box, do one of the following in the **Rotation** box:

    • Enter a positive value in degrees to rotate the macro attribute or label clockwise (CW).

    • Enter a negative value in degrees to rotate the macro attribute or label counter-clockwise (CCW).

    Tip

    • You can enter values that include decimals, such as 22.5°.

11. From the **Orientation** list, select a text box orientation if needed.

    Tip

    • In the vertical orientation, the software rotates the letters 45° CCW.

12. Click **Font** to open a dialog box where you can specify the text font, font style, size, effects, color, and script.

13. Click a blank area on the drawing sheet to position the text.

14. Click **File** > **Save** to save the drawing with the added redlining text.

Insert Redlining Graphical Elements

1. Open a single line diagram or a schematic drawing.

2. To activate redlining mode, click **Edit** > **Redlining**.

   ### 💡 Tip

   - When redlining mode is switched on, a check mark appears beside the menu command, and also, the text 'Redlining' appears as the label for the selected drawing sheet.

3. To insert a line, do one of the following:

   - Click **Edit** > **Insert** > **Line**.

   - On the toolbar, click ⟋ .

4. To insert a circle, do one of the following:

   - Click **Edit** > **Insert** > **Circle**.

   - On the toolbar, click ⊕ .

5. To insert a rectangle, do one of the following:

   - Click **Edit** > **Insert** > **Rectangle**.

   - On the toolbar, click ▢ .

6. Click on the drawing to position the redlining element.

7. Click **File > Save** to save the drawing with the added redlining items.

Edit Redlining Text

1. Open a single line diagram or a schematic drawing.

2. To activate redlining mode, click **Edit** > **Redlining**.

   ### 💡 Tip

   - When redlining mode is switched on, a check mark appears beside the menu command and also, the text 'Redlining' appears as the label for the selected drawing sheet.

3. Select the desired redlining text.

4. Do one of the following:

   - Click **Edit** > **Text**.

   - On the toolbar, click 𝐓 .

5. On the **Text Properties** dialog box, click the **General** tab.

6. Modify the annotation text in the **Caption** box.

    💡 **Tips**

    - You can type an unlimited number of lines with up to 28 characters in each line.

    - When you modify an annotation, the option **Set as watermark** is not available. You can set this option only when defining the text properties for the first time.

7. Click the **Properties** tab.

8. In the **Horizontal** and **Vertical** lists, select options to align text within the text box boundaries.

9. Select **Show border** if you want to display the text box border.

10. To rotate the annotation text box, do one of the following in the **Rotation** box:

    - Enter a positive value in degrees to rotate the macro attribute or label clockwise (CW).

    - Enter a negative value in degrees to rotate the macro attribute or label counter-clockwise (CCW).

    💡 **Tip**

    - You can enter values that include decimals, such as 22.5°.

11. From the **Orientation** list, select a text box orientation if needed.

    💡 **Tip**

    - In the vertical orientation, the software rotates the letters 45° CCW.

12. Click **Font** to open a dialog box where you can specify the text font, font style, size, effects, color, and script.

13. Click **File** > **Save**.

Related Topics
- *Delete Redlining*, page 362
- *Insert Redlining Graphical Elements*, page 361
- *Insert Redlining Text*, page 360
- *Managing Annotations and Redlining: An Overview*, page 356

Delete Redlining

1. Open a single line diagram or a schematic drawing.

2. Click **Edit** > **Redlining**.

3. Select a redlining item in the single line diagram.

   💡 **Tip**

   - You can select several redlining items at a time. To do so, hold the **Ctrl** key and click each item. The font and border color of the redlining items change when you select them.

4. Press the **Delete** key.

5. Click **File** > **Save**.

   📝 **Note**

   - When you delete redlining, it is not pasted to the Clipboard and you cannot reinsert it. You cannot restore a deleted redlining item.

Related Topics

Working with Drawing Items: An Overview

With SmartPlant Electrical, you can generate *schematics* and *single line diagrams (SLDs)*. After generation, you can view or manipulate the drawings before saving them. Among the options available to you are:

Viewing Drawings

Selecting Items

Using the Grid

Related Topics
- *Recommended Workflow for Generating Schematic Drawings*, page 271
- *Working with Schematics: An Overview*, page 270
- *Working with Single Line Diagrams Common Tasks*, page 294
- *Working with Single Line Diagrams: An Overview*, page 293

Selecting Drawing Items: An Overview

To select an item, you can use the **Select Tool** on the main toolbar. You can select motors, equipment, wires, labels, and so forth, and you can select item groups, assemblies, or embedded or linked objects.

Selecting Single Items

To select an item, click the **Select Tool** on the main toolbar.

When you click the **Select Tool**, the pointer changes to an arrow with a *locate zone* indicator at the end. As you pause on items in a drawing, the items appear in the highlight color. When an item is highlighted, you can click to select it.

When you select an individual item, the following things happen:

- The item changes to the selection color. You can change the selection color with the **Options** command on the **Tools** menu.

- The *handles* of the item appear if the item has handles. Handles are solid squares at significant positions on a selected item, such as end points and center points. Handles allow you to directly modify the item — drag a handle to change the shape of the item. Although you can select more than one item at a time, only one item can have handles at a time.

- If the item is *linked* or *embedded* into the current drawing, selecting it allows you to double-click it for editing.

When you select multiple items or grouped items, these change to the selection color.

Selecting Multiple Items

You can select more than one item at a time by clicking the **Select Tool** and then holding the **Shift** or **Ctrl** key as you click the items you want to select. Or, you can click the **Select Tool** and then drag to fence items.

You can use the **Select Tool** ribbon to choose if you want to select only items completely enclosed by the fence or any item that is partly enclosed by the fence. Many manipulation commands, like **Delete**, **Move**, **Copy**, and **Rotate**, can act upon all items in the select set.

Canceling the Selection of Items

To cancel selection of an item or group of items, you can click any empty point on the drawing.

Select an Item

1. On the main toolbar click the **Select Tool**.

2. Do one of the following:

    -   To select one item, click it.

    -   To select more than one item, hold the **Shift** or **Ctrl** key and click each item.

    -   To select more than one item at once, drag to fence the objects.

        💡 **Tip**

        -   To select all items in a drawing, right-click a blank area in the drawing and click **Select All** on the shortcut menu, or, to accomplish the same thing, press **Ctrl + A** or **Edit > Select All**.

    -   To select one of several overlapping items, use **PickQuick**. For more information, see Select an item using PickQuick.

3. To clear the selection of an item or items, do one of the following:

    -   Click in an empty portion of the drawing.

    -   Right-click in an empty portion of the drawing.

    -   Select another item without holding the **Shift** or **Ctrl** key.

    -   To clear the selection of one item and leave other items selected, click the item while holding the **Shift** or **Ctrl** key.

    📝 **Notes**

    -   When the **Select Tool** is active, selectable items are highlighted as you pass the pointer over them. When the item you want to select is highlighted, click to select it.

    -   Click **Tools > Options** to change the item highlight and selection colors and locate and break-away tolerances.

Related Topics
-   *Selecting Drawing Items: An Overview*, page 365

Using the Grid: An Overview

The grid and its grid lines allow you to place items in the drawing by aligning them with the grid lines or nearest intersection of the grid lines. You can use the grid if you want the items to line up at regular intervals in the drawing.

The grid is a set of imaginary lines in the drawing; they do not appear when you print the drawing. When you click **View > Snap Grid**, items always align with the grid lines or nearest intersection of the grid lines. You can view the grid by clicking **View > Show Grid**.

The grid lines for a dynamic grid move as you zoom in or out in a view. You can set dynamic grid lines to appear at fine, medium, or coarse levels. The grid lines appear at common major measurement increments. A dynamic grid displays index lines that intersect with the darker, solid grid lines.

A static grid displays solid grid lines that do not move as you zoom in or out. You can specify the increment settings on the **Grid** tab of the **View Properties** dialog box.

📝 Note

- You do not have to display the grid in order to snap items to it.

Related Topics
- *Selecting Drawing Items: An Overview*, page 365
- *Show Grid Command (View Menu)*, page 560
- *Snap Grid Command (View Menu)*, page 561
- *Viewing Drawings: An Overview*, page 368

Viewing Drawings: An Overview

When you generate or open schematics or single line diagrams, you can zoom and pan the diagram to view the areas of interest to you.

Zoom Area

To zoom in on a selected area, click **Zoom Area** on the main toolbar.

To select the area to zoom, drag over the area. You can redefine the zoom area position and dimensions by moving the mouse pointer while continuing to hold down the mouse button. For more information, see Zoom Area.

Zoom In

To zoom in on a selected area, click **Zoom In** on the main toolbar.

Click over the area where you want to zoom in. The point where you click becomes the center of the zoom-in area. For more information, see Zoom In.

Zoom Out

To zoom in on a selected area, click **Zoom Out** on the main toolbar.

Click over the area where you want to zoom out. The point where you click becomes the center of the zoom-out area. For more information, see Zoom Out.

Fit

To fit the drawing on the visible display area, click **Fit** on the main toolbar.

If you change the size of the display area, you must click the command again to make the drawing fit the modified display. For more information, see Fit.

Pan

To view different parts of the drawing in the visible display area, click **Pan** on the main toolbar.

You can move the drawing in the display area as required. For more information, see Pan.

Restore a Drawing View

After zooming or panning a drawing, you can restore the drawing to the previous view. Repeated selection of this option toggles between the previous and last views. For more information, see Restore a Drawing View.

Related Topics
- *Selecting Drawing Items: An Overview*, page 365
- *Using the Grid: An Overview*, page 367

Zoom In on an Area

1. On the main toolbar, click **Zoom Area**.

2. On the drawing view use the pointer to *fence* the area that you want to zoom in on. The selection then fills the view.

Notes

- To go back to the previous view, click **View > Previous**.

- To stop the repainting of items in the window, press **Esc** - this is convenient if you are working with a large drawing.

- To refresh the window, press **F5**.

- To quit the command, right-click or press **Esc**.

Related Topics
- *Viewing Drawings: An Overview*, page 368

Zoom In on the Drawing

1. On the main toolbar click **Zoom In**.

2. Click the view.

3. To quit the command, press **Esc** or right-click in the drawing.

Notes

- To go back to the previous view, click **View > Previous**.

- To stop the repainting of items in the window, press **Esc**. This is convenient if you are working with a large drawing. To refresh the window, press **F5**.

Related Topics
- *Viewing Drawings: An Overview*, page 368

Zoom Out from an Area

1. On the main toolbar, click **Zoom Out**.

   🔍

2. Click the drawing view.

📝 **Notes**

- To go back to the previous view, click **View > Previous**.

- To stop the repainting of items in the window, press **Esc**. This is convenient if you are working with a large drawing. To refresh the window, press **F5**.

- To quit the command, right-click or press **Esc**.

Related Topics
- *Viewing Drawings: An Overview*, page 368

Fit All Items in the Active Drawing View

- On the main toolbar, click **Fit**.

  🔲

Pan a Drawing View

1. On the main toolbar, click **Pan**.

   ✋

2. Click in the view you want to pan.
3. Click in the view again to define how far you want to pan.

📝 **Notes**

- You can click and drag the hand pointer to pan the view, too.

  ✋

- To go back to the previous view, click **View > Previous**.

- To stop the repainting of items in the window, press **Esc**. This is convenient if you are working with a large drawing. To refresh the window, press **F5**.

- To quit the command, press **Esc** or right-click in the drawing view.

Restore a Drawing View

- Click **View > Previous**.

📝 **Notes**

- You can stop the repainting of items in the window by pressing **Esc**. This is convenient if you are working with a large drawing.

- To refresh the window, press **F5**.

Related Topics
- *Viewing Drawings: An Overview*, page 368

View Associated Drawings of an Item

1. On the **Electrical Index,** select the item you want to view its associated drawings.

2. Right-click, and on the shortcut menu, click **Associated Drawings.**

3. On the **Associated Drawings** dialog box, from the **Drawing Type** list, select the drawing type you want to view.

4. Select a drawing from the **Associated Drawings** pane, and click **Open Drawing**.

📝 **Note**

- If the selected drawing type is a single line diagram, the software only shows the associated drawings of the selected item when the selected item is the main item of the single line drawing, and not a secondary item.

Working with Documents: An Overview

The software supports two types of documents - internal and external documents.

External documents are external files that you associate with electrical items in the software. These files can be reference documents or drawings, such as a general specification number, a requisition number, a vendor document, a civil or structural document, a site layout drawing, an electrical communications drawing, and so forth.

The software allows you to create a document reference repository that stores all the document references in your system. You can add external file references to your document repository or just create references that you can link to external files later. It is possible to display all the documents associated with a selected electrical item classified according to document category and type. Also, you can associate documents with multiple items if needed. Note that if you delete an electrical item that is associated with an external document, the software deletes the external document association automatically.

Internal documents are single line diagrams, electrical schematics, and power distribution board layout design drawings. You create these documents for selected electrical items and, therefore, you do not need to associate them. The software adds your internal documents to the document repository automatically and you can associate them with any electrical items as needed.

Related Topics
- *Working with External Documents Common Tasks*, page 378
- *Working with Internal Documents Common Tasks*, page 373

Working with Internal Documents Common Tasks

The following tasks are used frequently when you work with internal documents such as schematic drawings and single line diagrams. The tasks include managing revisions for single or multiple documents, archiving documents and comparing them with current versions.

Add a Document Revision

This procedure explains how to add a revision to a single document. For more information, see *Add a Document Revision*, page 374.

Manage Global Revisions

This option allows you to perform a number of revision activities for a group of selected documents. You can add, update, upgrade, and delete revisions in batch mode. For more information, see *Manage Global Revisions*, page 375.

Archive and Compare Documents

The software allows you to archive SLD and schematic drawing data in your SmartPlant Electrical database. This option then makes it possible to compare an archived version of a document with the data in the current version of that document. Before comparing an archived version of a document, you can click **View** to open it for viewing. To enable the archiving option, make sure that your Project Administrator switches on the archiving option in Options Manager. Once the archiving option is switched on, the software automatically archives every added or modified revision whether you did it for a single document or used the **Global Revision** feature for multiple documents. However, the software does not allow you to archive miscellaneous drawings. For more information on comparing an archived version of a document with its current data, see *Compare an Archived Document with a Current Document*, page 376.

Include Revision Data in Registered Reports

This procedure explains how to include revision data in registered reports. You can display as many revision levels as you require. For more information, see *Include Revision Data in Registered Reports*, page 413.

Add a Document Revision

1. Open or generate an SLD or a schematic drawing.

2. Click **Edit** > **Document Properties**.

   💡 **Tip**

   - To access the **Document Properties** command, first select an item in the drawing.

3. On the **Document Properties** dialog box, click **New**.

4. Under **Revision method**, select a revision numbering method that you require for the current document.

   💡 **Tip**

   - If a document you are revising already has a revision, do not change the revision method unless you want the software to start a new revision numbering.

5. In the **Revisions** group box, under **Revision Numbering**, accept the displayed value or type the one that you require.

   💡 **Tips**

   - If you type a value that does not comply with the current revision method, the next time you add a new revision, the software starts a new numbering. For example, the selected revision method is P0, P1, P2 and the last revision is P3. If you add a new revision number RRR and then another one using the P0, P1, P2 method, the software displays the newly added revision as P0.

   - If you type a value that complies with the numbering method but the value is not sequential, the software increments the number of the next revision that you add. For example, the selected revision method is P0, P1, P2 and the last revision is P3. If you add a new revision number P1, the software will increment the following revision to P2.

6. Type all the other pertinent information relating to the revision date, person who created and checked the revision, and so forth.

📝 **Note**

- You can delete a selected revision by highlighting a row and clicking **Delete**. The software updated the revision numbers automatically if the subsequent revision number complies with the selected revision method.

Related Topics
- *Compare an Archived Document with a Current Document*, page 376
- *Manage Global Revisions*, page 375

Manage Global Revisions

1. In the list view of the **Electrical Index**, select the documents that you require.

2. Click **Actions > Global Revisions**.

3. From the **Activity** list, select an activity to perform with a global revision:

   - **Add revision** - adds a new revision to the selected documents in accordance with the value that you specify for the **Revision number** setting. The software determines which revision method suits best the value that you typed in the **Revision number** field and adjusts the revision method accordingly. If the revision number value that you are adding conforms with method of the last existing revision, the software will continue to number the new revision accordingly. If the software cannot determine a suitable revision method for the typed value, the revision method will switch to **Other**. Also, if you do not enter a value in the **Revision number** field, the software will add a new revision using the **Other** method and will not provide a value in the **Revision number** field of the individual documents.

   - **Upgrade revision** - adds a new revision by raising the last revision number of all the selected documents. The software identifies the revision numbering method of each document (P0, P1, P2; 0,1,2, A,B,C, and so forth) and raises the revision number to the next value according to the existing revision method. You cannot change the revision method but you can update all the other settings by entering new settings.

   - **Delete last revision** - deletes the last revision of all the selected documents.

   - **Delete all revisions** - deletes all the revisions of all the selected documents.

4. From **Revision method for items without revisions**, select a revision method if some of the selected documents have not been revised yet.

5. Edit the settings as you require.

Note

   - The software archives all the added and upgraded revisions of the selected documents so that you can compare the data of an archived document with the current version. For more information, see *Compare an Archived Document with a Current Document*, page 376.

Related Topics

   - *Add a Document Revision*, page 374
   - *Compare an Archived Document with a Current Document*, page 376
   - *Working with Internal Documents Common Tasks*, page 373

Compare an Archived Document with a Current Document

1. Open or generate an SLD or a schematic drawing.

2. Modify the drawing as you require.

3. Save the drawing.

4. Click **Edit** > **Document Properties**.

5. Add a revision and click **OK** on the **Document Properties** dialog box.

   💡 **Tip**

   - Repeat the above procedure as many times as required to add archived versions of the document to the database.

6. In the **Electrical Index**, expand the **Documents** folder hierarchy to display the names of the existing SLDs or schematics.

7. Under the **Single Line Diagrams** or **Schematic Drawings** folder, select the document you require and do one of the following to open the **Compare Documents** dialog box:

   - Click **Actions** > **Compare Documents**.

   - Right-click the document and select **Compare Documents** from the short-cut menu.

8. Select an archived version of the document and click **Compare**.

   💡 **Tip**

   - Before clicking **Compare**, you can click **View** to open the selected drawing that you want to compare. The software opens the archived drawing in view mode only.

📝 **Notes**

- In SLDs and schematics, the software displays clouds around the items that differ from the current data. The last revision number is shown in a triangle beside the cloud.

- The software uses index numbers to identify the changes in associations between electrical items.

- The software generates a tabular report where it shows all the changes and shows the changes in associations using index numbers as they appear in the drawing.

- For registered reports, the software opens two Excel files. One of the Excel files displays changed data with a blue shading. The second file is a summary of all changes and it is called **Registered Comparison Report**. This report displays the previous and current data for each tag that has undergone a change. Note that you can compare a registered report only if this is a simple tabular report.

- In SLDs, the software indicates the following changes:

- All added and deleted electrical items except for control stations.

- Added, deleted, and updated properties of electrical items.

- Changes in associations.

- Attachment to a different document template.

- In schematic drawings, the software indicates the following changes:

- Added, deleted, and updated properties of electrical items.

Related Topics
- *Add a Document Revision*, page 374
- *Manage Global Revisions*, page 375
- *Working with Internal Documents Common Tasks*, page 373

Batch Compare Documents

1. In the **Electrical Index**, expand the **Documents** folder hierarchy to display the names of the existing SLDs or schematics.

2. Select the **Single Line Diagrams** or **Schematic Drawings** folder.

3. In the lower pane, select the documents you require and do one of the following to open the **Batch Compare Documents** dialog box:

   - Click **Actions** > **Compare Documents**.

   - Right-click the document and select **Compare Documents** from the short-cut menu.

4. To save the comparison report, select **Save files**.

5. Under **Path**, type the folder path where you want to save the report, and from the **Save as type** list, select the desired file format.

6. To print the current and last revised versions of the selected drawings, select **Print drawings**.

7. To print a set of reports comparing the changes between the current and last revised drawings, select **Print comparison reports**.

Notes

- In batch mode, the software compares the current data with the last saved revision. To compare two archived documents, select a single document only and use the **Compare Documents** command.

- In SLDs and schematics, the software displays clouds around the items that differ from the current data.

- The software uses index numbers to identify the changes in associations between electrical items.

- The software generates a tabular report where it shows all the changes and shows the changes in associations using index numbers as they appear in the drawing.

- In SLDs, the software indicates the following changes:

- All added and deleted electrical items except for control stations.

- Added, deleted, and updated properties of electrical items.

- Changes in associations.

- Attachment to a different document template.

- In schematic drawings, the software indicates the following changes:

- Added, deleted, and updated properties of electrical items.

Related Topics

- *Add a Document Revision*, page 374
- *Compare an Archived Document with a Current Document*, page 376
- *Manage Global Revisions*, page 375
- *Working with Internal Documents Common Tasks*, page 373

Working with External Documents Common Tasks

The following tasks are used frequently when defining external document references and associating external documents with electrical items.

Define an External Document Reference

You define a document reference by adding a reference to the list of document references. You can then associate this reference with an external file. The software adds this association by storing the file path in the database. For more information, see *Define an External Document Reference*, page 380.

Associate External Documents with Electrical Items

After you define external document references, you select one or more electrical items and associate them with external documents. You can associate an electrical item with a document reference or with an external file that is linked to the document reference. The software allows you to associate single or multiple electrical items with the same external document. Also, you can associate the same external file with multiple items in the software. For more information, see *Associate External Documents with Electrical Items*, page 381.

Dissociate an External Document

This option allows you to dissociate electrical items from external documents. For more information, see *Dissociate an External Document*, page 382.

Dissociate an External Document from Multiple Electrical Items in Batch Mode

This option allows you to dissociate an external document from multiple electrical items in batch mode. For more information, see *Dissociate an External Document from Multiple Electrical Items in Batch Mode*, page 383.

Open an External Document

This option allows you to open an external file that is associated with a document reference. For more information, see *Open an External Document*, page 383.

Insert External Graphics Files into SmartPlant Electrical Templates

This option allows you to insert an external graphics file into a SmartPlant Electrical template. An external graphics file can be any file created in SmartSketch or any other CAD application. For more information, see *Inserting External Graphics Files into SmartPlant Electrical Templates*, page 384.

Define an External Document Reference

1. Click **Actions** > **Define Document Reference**.

2. On the **Document Reference Definition** dialog box, under **Filter by**, select a document category.

3. Select a document type.

4. Click **Add**.

5. Under **Edit**, type a value in the **Document** box.

   💡 **Tip**

   - If you selected **All types** from the **Document type** list under **Filter by**, select the desired value from the **Document type** list under **Edit**.

6. Type a suitable document description in the **Description** box if needed.

7. Beside the **Path and file name** box, click the ellipsis button ⬚ to select the desired file.

   💡 **Tip**

   - You do not have to associate a document reference with an external file. You can do this at some other time when needed.

8. On the **Select File** dialog box, select the desired file.

9. Click **Apply** to store the document reference in the database and select another document category if needed.

Related Topics
   - *Associate External Documents with Electrical Items*, page 381
   - *Working with Documents: An Overview*, page 372
   - *Working with External Documents Common Tasks*, page 378

Associate External Documents with Electrical Items

1. Define a reference for the document that you want to associate. For details, see *Define an External Document Reference*, page 380.

2. In the **Electrical Index**, select an electrical item.

   💡 **Tip**

   - You can also associate multiple electrical items with the same external document. In the **Electrical Index**, click a folder that contains electrical items and highlight in the list view pane the items that you require.

3. Click **Actions** > **Associate Document**.

4. On the **Associate Documents** dialog box, under **Filter by**, select a document category and type from the lists.

5. Click **Add**.

6. On the **Find** dialog box, click **Find Now** to display the available documents.

   💡 **Tip**

   - The **Find** dialog box finds only those documents that belong to the document category and type that you select on the **Associate Documents** dialog box.

7. Select each document that you want to associate with the selected item and then click **Apply**.

   💡 **Tip**

   - To associate documents belonging to another category or type, return to the **Associate Documents** dialog box, select the desired document category and type, click **Associate** again, and repeat the succeeding steps in this procedure.

Related Topics
- *Define an External Document Reference*, page 380
- *Dissociate an External Document*, page 382
- *Working with Documents: An Overview*, page 372
- *Working with External Documents Common Tasks*, page 378

Dissociate an External Document

1. In the **Electrical Index**, select an electrical item.

2. Click **Actions** > **Associate Document**.

3. On the **Associate Documents** dialog box, under **Filter by**, select a document category.

4. From the **Document type** list, select a document type.

5. Under **List**, highlight the document that you want to dissociate.

6. Click **Remove**.

    💡 **Tip**

    *   To dissociate a document belonging to another type or category, click **Apply** and then select another document type and category as required.

Related Topics
*   *Define an External Document Reference*, page 380
*   *Dissociate an External Document from Multiple Electrical Items in Batch Mode*, page 383
*   *Working with Documents: An Overview*, page 372
*   *Working with External Documents Common Tasks*, page 378

Dissociate an External Document from Multiple Electrical Items in Batch Mode

1. In the list view of the **Electrical Index**, select the electrical items you require.

2. Click **Actions** > **Associate Document**.

3. On the **Associate Documents** dialog box, under **Filter by**, select a document category.

4. From the **Document type** list, select a document type.

5. Click **Dissociate**.

6. On the **Find** dialog box, click **Find Now**.

7. Select the document that you want to dissociate from the selected items and click **OK**.

8. On the **Associate Documents** dialog box, click **OK** or **Apply**.

Related Topics
*   *Define an External Document Reference*, page 380
*   *Working with Documents: An Overview*, page 372
*   *Working with External Documents Common Tasks*, page 378

Open an External Document

1. In the **Electrical Index**, select an electrical item.

2. Click **Actions** > **Associate Document**.

   💡 **Tip**

   - You can also open an associated document by clicking **Actions** > **Define Document Reference**.

3. On the **Associate Documents** dialog box or the **Document Reference Definition** dialog box, under **List**, select the document that you want to open.

   💡 **Tip**

   - On the **Document Reference Definition** dialog box, under **Filter by**, select the required document category and type.

4. Click **Open**.

Related Topics

Inserting External Graphics Files into SmartPlant Electrical Templates

The software allows you to create project drawings based on graphics files created in SmartSketch or any other CAD application. These project drawings are called miscellaneous drawings. You create miscellaneous drawings by inserting an external graphics file into a SmartPlant Electrical template.

Once you create a miscellaneous drawing and save it to your SmartPlant Electrical database, the software updates the drawing title block and the revision data. The software also enters the revision property macros, which enables the retrieval of information from the database when you reopen the drawing. SmartPlant Electrical organizes these drawings in the **Miscellaneous Drawings** folder in the **Electrical Index**.

You can open a miscellaneous drawing and manage its revisions as you require. However, you cannot archive a miscellaneous drawing and therefore you cannot compare it with a previous revision.

Just like with external documents, SmartPlant Electrical allows you to associate a miscellaneous drawing with an electrical item.

Related Topics
- *Insert an External Graphics File into a SmartPlant Electrical Template*, page 385
- *Working with External Documents Common Tasks*, page 378

Insert an External Graphics File into a SmartPlant Electrical Template

1. Open Options Manager and set a default path for miscellaneous drawings. For details, see in the Options Manager User's Guide: *Define File Locations*.

2. In SmartPlant Electrical, double-click the **Documents** folder and then right-click the **Miscellaneous Drawings** folder in the **Electrical Index**.

3. On the shortcut menu, click **New Miscellaneous Drawing**.

4. On the **New Miscellaneous Drawing** dialog box, beside the **Template** box, click the ellipsis button to select a template that you require.

5. Under **Document number**, type a document name. Note that this is a required field.

6. Type a short description if needed and click **OK**.

7. In the drawing that opens, to insert a symbol, click **Edit > Insert > Symbol**.

8. On the **Select Symbol File** dialog box, select a symbol and click **OK**.

9. To place the symbol in the drawing, position the cursor in the open drawing and then press the left mouse button. Note that the cursor marks the symbol origin (the left bottom corner).

   Tip

   • You can insert as many symbols as you require.

10. Click **File > Save** to save the miscellaneous drawing.

Notes

• After saving a miscellaneous document, you can set the appropriate revisions as you require. For details, see *Add a Document Revision*, page 374.

• You can associate an electrical item with a miscellaneous drawing. For details, see *Associate External Documents with Electrical Items*, page 381.

Related Topics

• *Inserting External Graphics Files into SmartPlant Electrical Templates*, page 384
• *Working with External Documents Common Tasks*, page 378

Generating Reports: An Overview

Reporting is the process of retrieving information from the database and displaying the information as formatted output. At any time during the design creation process, you can create a report. Each report consists of a Microsoft Excel workbook and a report definition, which describes the data to collect and how to organize the data in the workbook.

Each report you create is based on an item type. This item type serves as the starting point for collecting data for your report. Examples of item types include motors, cables, control stations, and buses. Several default report templates already exist; however, reports are fully customizable. You can create your own reports that contain the information you want to see in a format you choose.

The relationships that exist between the various item types constitute additional available information for a report. For example, a motor is related to the power distribution board and bus with which it is associated. When creating your report, only items that have a relationship with your selected item type can be used as input.

Your report definition contains one or more report items organized in a tree hierarchy. Each report item is based on a main item type. Each report contains at least one report item to define the item type of the report. For example, a report based on the load item type contains a report item named **Load**. This report item makes the properties associated with each load available to be included in your report; however, you do not have to include every available property in your report if it is not appropriate. You can define additional report items to access more properties for more item types. For example, a load report item can be added to access data about motors because motors and equipment are related.

The location of a report item in the tree hierarchy affects which properties are collected for the associated item type. For example, if motor is added as the top-level item in the tree, all motors in the database are collected for your report. If the motor is added as a child of "Equipment:", only the information about motors is collected.

Some SmartPlant Electrical reports are very simple and some are very complex. Therefore, you need to be well acquainted with the SmartPlant Electrical Data Model to be able to generate meaningful reports. All the available items and properties are based on the associations made in the Data Model. You may need to select either a single item type or use a chain of item types, depending on your needs.

You begin your reporting process by selecting a report template from the **Reports** menu and then selecting items and properties to be included in your report. Then the software performs the following tasks:

1. Microsoft Excel starts. The report template is copied to the report output folder, and then the Excel workbook opens.

2. Your report definition is retrieved from the Excel workbook.

3. Your data is retrieved based on the report item definitions of the report template.

4. Data prints to the Microsoft Excel workbook using the cell mapping data in your report definition.

✎ Notes

- You must have installed Microsoft Excel on your computer to display reports.

- The software stores the reports that you generate in your Windows directory under \Profiles\<username>\My Reports\Output.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Create a Report Template Based on an Existing Template*, page 419
- *Display the Properties of a Report Template*, page 429
- *Edit a Report Template*, page 431
- *Generate a Report*, page 387

Generate a Report

1. Either select the items from the Tabular Editor that you want to include in the report, or select no items and the software will give you the option to report on all items in step 4.

2. Click **Reports > Plant Reports**.

💡 Tip

- Or click **Reports > My Reports** to select a customized report that you defined already.

3. Click the type of report you want to generate.

4. On the **Plant Reports** dialog box or the **My Reports** dialog box, choose the items you want to report on in the **Report using** area.

5. Click **OK**.

✎ Notes

- You must have installed Microsoft Excel on your computer to display reports.

- The software stores the reports that you generate on your local computer in the Windows directory under \Profiles\<username>\My Reports\Output.

Related Topics
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Generate a Report for Selected Items*, page 388
- *Generating Reports: An Overview*, page 386

Generate a Report for Selected Items

1. In the **Electrical Index**, select the items for which you want to generate a report.

2. Click **Reports > Plant Reports**.

   💡 **Tip**

   - Alternatively, click **Reports > My Reports** to select a customized report that you defined already.

3. On the **Plant Reports** dialog box or the **My Reports** dialog box, choose the items you want to report on.

4. In the **Report using** group box, select one of the following options:

   - **Current selection** - Produces a report containing the selected items in your current table. This option is not available if no table items are selected.

   - **Entire table** - Produces a report containing the contents of the Tabular Editor.

5. Click **OK**.

📝 **Notes**

- You must have installed Microsoft Excel on your computer to display reports.

- The software stores the reports that you generate on your local computer in the Windows directory under \Profiles\<username>\My Reports\Output.

Related Topics

Shipped Reports

SmartPlant Electrical provides you with a number of predefined reports. You can use these reports as templates for any other reports. You can define your own sort and filter settings. Also, you can use sort, filter or other queries to create new reports.

Note that if you do not select particular items in a table that you open in the Tabular Editor, the software generates a report that includes all the tags belonging to the item type that the report is based on.

Every time you generate a report, the software retrieves the relevant data from the database and calculates the pertinent results based on the currently retrieved values. The software does not store the report and the calculation results in the database, but rather in the Excel file that the software creates after generating the report. The software recalculates the values every time you generate the report. Therefore, the report displays the data only as it exists at the time of the report generation. You can then save the report as an external Excel file if needed.

Note that there are hidden columns and Visual Basic code in the templates of these reports. Be very careful when editing a template or a report, as this may render your report unusable.

The following reports are available:

I/O Signal List

I/O Signal List is a report that is intended for publishing signal data to SmartPlant Instrumentation. The report includes all I/O signals defined in the units of the current plant. This report provides the following information:

- I/O signal tags
- Description
- I/O types
- Host
- Card
- Channel
- Address
- CS tag
- Reference instrument loop

I/O Signal List - with issue

I/O Signal List - with issue is a report that is intended for publishing signal data to the integrated environment. The report includes all I/O signals defined in the units of the current plant. This report provides the following information:

- I/O signal tags
- Description
- I/O types
- Host
- Card
- Channel
- Address
- CS tag
- Reference instrument loop

I/O Signal and Equipment - with issue

I/O Signal and Equipment - with issue is a report that is intended for publishing signal and equipment data, or just equipment data to the integrated environment. The report includes all I/O signals and equipment defined in the units of the current plant. The report provides the following information:

- I/O signal tags
- Description
- Equipment tags
- Item type
- I/O type
- Host
- Card
- Channel
- Address
- CS tag
- Reference instrument loop

Instrument and Cabinet Load List

Instrument and Cabinet Load List is a report that is intended for publishing instruments and cabinets that have been retrieved from SmartPlant Instrumentation as loads. The report actually displays data for all electrical equipment types; however, when you use it for publishing data to SmartPlant Instrumentation, the software automatically selects data for instruments and cabinets only; you should not use a filter to do so. This report provides the following information:

- Instrument / cabinet tags
- Tag types
- Description
- Instrument rated power
- Cabinet rated power

Instruments and Cabinet Load List - with issue

Instrument and Cabinet Load List is a report that is intended for publishing instruments and cabinets that have been retrieved from SmartPlant Instrumentation as loads to the integrated environment. The report actually displays data for all electrical equipment types; however, when you use it for publishing data to the integrated environment, the software automatically selects data for instruments and cabinets only; you should not use a filter to do so. This report provides the following information:

- Instrument / cabinet tags
- Tag types
- Description
- Instrument rated power
- Cabinet rated power

Electrical Load List

Electrical Load List is a report that you generate based on the load item type. The software retrieves the load tags with the relevant load tag data, such a rated power, brake power for motors, as well as values for efficiency, power factor, full load current, and so forth.

PDB Load Summary

PDB Load Summary is a report that you generate based on a load item type. The purpose of this report is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual coincidence factors, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver.

The report specifics are as follows:

- For each associated load, the report lists such properties as load rated power, the load demand factor, operating mode, and the electrical rated values (in kW and kVAr) of the load.

- On the first page of the report, the software shows the loads that are not yet associated with any PDB. On the subsequent pages of the report, the software lists each PDB and its associated loads.

- The report includes only those PDBs that are associated with loads. However, the report does not include loads that do not belong to the load item type.

- Loads that are connected to circuits for which the circuit mode value is **Disconnected** appear in load summaries, but the software does not take those loads into account in the total load calculation.

- The report groups and sorts the loads according to their associated PDB.

- The software sums the electrical quantities to provide the total running load and the total peak load based on the coincidence factors that has been defined for each load.

- Under each PDB in the report, the software sorts the loads according to their bus association and then alphanumerically under the bus.

- The software does not filter out any specific PDB. If you do not filter the PDBs, the software lists all the existing PDBs. To create a report for a specific PDB, use an appropriate filter. The software displays each PDB on a separate page. The first page is reserved for those loads that are not associated with a PDB.

- The report multiplies each electrical load by its coincidence factor (x, y, z or zz).

- Based on the total kW and KVA values, the software calculates the needed power factor in the report.

- The software does not store the report and the calculation results in the database, but rather in the Excel file that the software creates after generating the report. The software recalculates the values every time you generate the report. Therefore, the report displays the data only as it exists at the time of the report generation. You can then save the report as an external Excel file if needed.

- For associated capacitors, the software subtracts the rated kVAr value of the capacitor from the total connected rated kVAr value. Therefore, the rated electrical power (kVAr) may display a negative value. This means that there is a lagging power factor.

- When localizing the software, do not translate the values (**Continuous**, **Intermittent**, **Standby**, and **Spare**) that appear in the **Operating Mode** list of the Data Dictionary. The software will not be able to perform some of the calculations if these values are changed.

PDB Load Summary - Consumed Power Values

The PDB Load Summary - Consumed Power values report displays consumed power values instead of rated power values. The rest of the report is identical to the PDB Load Summary report.

All Feeder Load Summary Report

The All Feeder Load Summary report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Tabular Editor**.

Apart from one major function, the All Feeder Load Summary report has the same purpose as the PDB Load Summary report which is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual coincidence factors, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. While the PDB Load Summary report calculates **only** the total direct load connected to a bus, the All Feeder Load Summary report **also** sums the loads that are connected to downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

For a full explanation of this report, see *All Feeder Load Summary Report*, page 391.

All Feeder Load Summary Consumed Power Report

The All Feeder Load Summary - Consumed Power Values report displays consumed power values instead of rated power values. The rest of the report is identical to the All Feeder Load Summary report. For full explanation, see *All Feeder Load Summary Consumed Power Report*, page 400.

All Feeder Load Summary (Enhanced) Report

The All Feeder Load Summary (Enhanced) report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Electrical Engineer**. This report is based on either the consumed or rated power of the selected loads.

The All Feeder Load Summary (Enhanced) report provides a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the coincidence factors of the relevant buses or the individual loads, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. The enhanced report sums the loads that are connected to the downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

For full explanation, see *All Feeder Load Summary (Enhanced) Report*, page 404.

PDB Load Summary by Operating Mode Report

The PDB Load Summary by Operating Mode report is intended to complement the All Feeder Load Summary report, which calculates the running and peak loads according the coincidence factor of each load or according to the bus coincidence factor. This report satisfies the need to calculate a load summary according to special operating modes, such as emergency or life support systems for offshore platforms. The software provides ten operating mode factors (duty and load factors) for each load or bus. The All Feeder Load Summary by Operating Mode report displays the calculated figures for the total power according to each of the ten operating modes. Note that the output of the All Feeder Load Summary by Operating Mode report is affected by your settings on the **Reports** tab of the **Preferences** dialog box exactly in the same way as the All Feeder Load Summary report.

For a full explanation of this report, see *PDB Load Summary by Operating Mode Report*, page 408.

Cable Take-Off

Cable Take-Off is a report that provides a summary of all the existing project cable types, including the unsized cables.

The report specifics are as follows:

- Provides technical details about the cable types.
- Calculates the total estimated and design lengths of each cable type.
- Calculates the lengths of single core conductor configuration cables based on the number of conductors of each cable.
- Provides estimated cable lengths both in meters and feet.

- Allows you to expand each cable type to view the individual cable tags.

Cable Schedule

Cable Schedule is a report that lists all the existing project cables.

This report provides the following information:

- Cable tag names
- Cable types
- Estimated and design cable lengths
- Store part numbers
- Cable origin and destination sides (the **To** and **From** cable sides). The **To** and **From** information includes the tag of the electrical item that the cable is associated with. If one of the cable sides is assigned to a PDB, the report displays the path of the circuit tag: PDB/ bus/(cell)/ circuit.

Cable Schedule - with issue

Cable Schedule - with issue is a report that lists all the existing project cables for publishing to the integrated environment.

This report provides the following information:

- Cable tag names
- Cable types
- Estimated and design cable lengths
- Store part numbers
- Cable origin and destination sides (the **To** and **From** cable sides). The **To** and **From** information includes the tag of the electrical item that the cable is associated with. If one of the cable sides is assigned to a PDB, the report displays the path of the circuit tag: PDB/ bus/(cell)/ circuit.

Cable Sizing Summary

Cable Sizing Summary is a report that lists all the cable sizing input and output data for a specified power cable. You are required to create a filter for this report to display power cables only. This report is useful as a quick reference for your sizing considerations and criteria at any time during the life cycle of your project.

This report provides the following information:

- Cable tag name.

- Cable origin and destination sides (the **To** and **From** cable sides). The **To** and **From** information includes the tag of the electrical item that the cable is assigned to.

- Input values such as load voltage, full load current, number of phases, and so forth.

- Sizing results such as reference cable, conductor size, calculated voltage drop, and so forth.

Cable Wiring Schedule

Cable Wiring Schedule is a report that provides a list of selected cables and their conductors showing to which terminal strips and terminals the cables are connected. The report is based on the item type of the cables for which you want to generate a report. When customizing your cable wiring schedule report, use the following conductor properties to define the association between the conductors and the terminals that are connected on both sides of the cables.

- Side1.Terminal.Item.Tag

- Side1.Terminal Strip.Item.Tag

- Side2.Terminal.Item.Tag

- Side2.Terminal Strip.Item.Tag

Terminal Strip Schedule Reports

SmartPlant Electrical provides with three terminal strip schedule reports. These reports show wiring information from the perspective of the wiring equipment that you select in the **Electrical Index**. The connection between the terminals and the conductors is based on the Conductor-Terminal table. The following reports are available:

- Terminal Strip Schedule — A report based on the terminal strips you select in the **Electrical Index**.

- Terminal Strip Schedule - Panel — A report based on the panels you select in the **Electrical Index**.

- Terminal Strip Schedule - PDB — A report based on the power distribution boards you select in the **Electrical Index**.

Drum Composition

Displays the cable assignments for each drum in your plant.

Segment Width Estimate

You can use the Segment Width Estimate report for calculating the tray widths and number of layers needed so that you can select appropriate reference cableway components.

Cableway Component Material Take-Off

Displays the quantity of items required for each type of reference cableway component when ordering items from the manufacturer.

Cableway Segment Schedule

Displays the list of cables in each cableway segment that makes up your cable routing.

Segment Fill

Displays the fill status and tray loading for each segment and indicates whether or not the segment is overfilled.

PDB Schedule

PDB Schedule is a report that you generate based on the circuit item type. The report specifics are as follows:

- The report lists, for all the power distribution boards in the current plant, all the circuits, loads, circuit internal components with the technical details of each circuit.

- The report includes the associated schematic drawings.

- The report uses a predefined filter to display the associated schematic drawings.

Power Distribution Panel Schedule

PDB Schedule is a report that is intended for single phase loads connected to distribution panels. The report specifics are as follows:

- The report lists, for all the power distribution boards in the current plant, all the circuits, each with its protection device type and current rating (the software supports a single protection device per circuit) and the loads associated with each circuit.

- The report sorts the circuits by phase and then by circuit (all the circuits for phase 1, then phase 2, and then phase 3).

- The report calculates the following: Full load current for each phase, Total load current of the bus, Total kW per phase, Total kW per bus.

- This report is not suitable for PDBs that include 3-phase loads.

All Feeder Load Summary Report

The All Feeder Load Summary report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Tabular Editor**. This report is based on the rated power of the selected loads.

Apart from one major function, the All Feeder Load Summary report has the same purpose as the PDB Load Summary report which is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual coincidence factors, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. While the PDB Load Summary report calculates **only** the total direct load connected to a bus, the All Feeder Load Summary report **also** sums the loads that are connected to downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

Once you start generating an All Feeder Load Summary report (for details, see *Generate a Report for Selected Items*, page 388), the software starts retrieving the data of all the loads that are connected to the buses that belong to the PDB that you selected in the **Electrical Index**. The software retrieves the values for one bus at a time (the summary is per bus within the selected PDB) including the loads that are connected to the downstream loads that are connected and fed from the bus for which the software calculates the summary. The generated report shows the individual loads connected to the bus as well as all the feeders that feed the downstream buses and their loads.

💡 Tip

- Note that circuits have a property called **CircuitMode** with values **Connected** and **Disconnected**. If you set this property to **Disconnected**, the drill down algorithm will stop the calculation and the software will not roll up the loads connected to this feeder.

The output of the All Feeder Load Summary report depends on the settings that you make in the Options Manager or on the **Reports** tab of the **Preferences** dialog box. There are two major factors that influence the results of the report:

- The inclusion of coupled buses in the All Feeder Load Summary report.

- The calculation method for the All Feeder Load Summary report.

The following table shows how the inclusion of coupled buses affects the output of the All Feeder Load Summary report. These settings define how the software handles the buses that are coupled with the selected bus and the downstream buses.

Preference Setting	Result
Do not include coupled buses	The software disregards any existing tied bus on the selected PDB. The software also disregards all the buses that are not powered directly from a feeder in the power path. The software generates a summary that is based on all the loads that are connected directly to the current bus and the loads that are connected to the downstream buses that are connected to the current bus. The software excludes all the loads on the redundant or coupled buses as well as all the transformers and converting equipment.
Include coupled buses connected by couplers only	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "coupler circuits". Therefore, all the loads on tied buses are also included in the calculation if the tied bus is connected to the selected bus using a coupler.
Include coupled buses connected by couplers and risers	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "riser circuits". This is the most inclusive option that allows you to include in the calculation not only the loads on tied buses connected by couplers but also those that are connected by riser circuits.
Account for converting equipment power losses	Instructs the software to account for power losses when using converting equipment.

The second major factor that affects the results of the report is the calculation method for the All Feeder Load Summary report.

Selecting the **Use each load coincidence factor** option instructs the software to look for the load operating mode (continuous, intermittent, spare, or standby). The software multiplies the rated kW and kVAr by the respective coincidence factor (X, Y, Z, ZZ) and uses these new "factored" numbers in the summary. The software calculates two sets of load summaries - the total load for **normal** and **peak** operations.

The calculated total load for **normal** operation is a summary of all the loads whose operating mode is continuous and intermittent. The summary does not include any standby or spare loads. The software calculates this summary based on the following equation:

Sigma load kW = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number]

Sigma load kVAr = Sigma of the [each of the continuous loads kVAr multiplied by their individual X coincidence number + each of the intermittent loads kVAr multiplied by their individual Y coincidence number]

Sigma kVA = Sqrt [Sigma load kW+ Sigma load kVAr]

Average load power factor = Sigma KW/Sigma kVA

The calculated total load for **peak** operation is a summary of all the loads whose operating mode is continuous, intermittent, standby, or spare. The software calculates this summary based on the following equation:

Sigma load kW = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number + each of the standby loads KW multiplied by their individual Z coincidence number + each of the spare loads kW multiplied by their individual ZZ coincidence number]

Sigma load kVAr = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number + each of the standby loads kW multiplied by their individual Z coincidence number + each of the spare loads kW multiplied by their individual ZZ coincidence number]

Sigma kVA = Sqrt [Sigma load kW + Sigma load kVAR]

Average load power factor = Sigma kW/Sigma kVA

The **second option** setting for calculating the load summary uses average bus coincidence factors instead of the individual coincidence factors of the loads. The only difference is that in this case, instead of multiplying the loads by their individual coincidence factors, the software multiplies all the loads of a specific bus by the bus X, Y, Z, or ZZ. The equation for this option is as follows:

Sigma load kW = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number]

Sigma load kVAr = Sigma of the [all of the continuous loads kVAr multiplied by the bus X coincidence number + all of the intermittent loads kVAr multiplied by the bus Y coincidence number]

Sigma kVA = Sqrt [Sigma load kW + Sigma load kVAr]

Average load power factor = Sigma KW/Sigma kVA

The total load for peak operation in this case also includes continuous, intermittent, standby, and spare loads. The equation for this option is as follows:

Sigma load KW = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number + all of the standby loads kW multiplied by the bus Z coincidence number + all of the spare loads kW multiplied by the bus ZZ coincidence number]
Sigma load kVAr = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number + all of the standby loads kW multiplied by the bus Z coincidence number + all of the spare loads kW multiplied by the bus ZZ coincidence number]
Sigma kVA = Sqrt [Sigma load kW+ Sigma load kVAr]
Average load power factor = Sigma kW/Sigma kVA

In addition to the two options mentioned above, the software also allows you to specify a bus for which you can perform a calculation based on the bus rated values instead of performing a roll up calculation. To make such a calculation, in the **Properties** window of the pertinent bus, set the **Drill Down Flag** property to **False**. This setting instructs the software to calculate the load based on the bus rated power values of the selected bus.

At the bottom of the All Feeder Load Summary report, the software displays the calculated total of all the loads of the selected PDB. The software sums the kW and kVAr values of all the buses and also calculates the kVA and the power factor.

Related Topics
- *Generate a Report for Selected Items*, page 388
- *PDB Load Summary by Operating Mode Report*, page 408
- *Shipped Reports*, page 390
- *Working with the Electrical Index: An Overview*, page 44

All Feeder Load Summary Consumed Power Report

The All Feeder Load Summary report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Tabular Editor**. This report is based on the consumed power of the selected loads.

Apart from one major function, the All Feeder Load Summary report has the same purpose as the PDB Load Summary report which is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual coincidence factors, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. While the PDB Load Summary report calculates **only** the total direct load connected to a bus, the All Feeder Load Summary report **also** sums the loads that are connected to downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

Once you start generating an All Feeder Load Summary report (for details, see *Generate a Report for Selected Items*, page 388), the software starts retrieving the data of all the loads that are connected to the buses that belong to the PDB that you selected in the **Electrical Index**. The software retrieves the values for one bus at a time (the summary is per bus within the selected PDB) including the loads that are connected to the downstream loads that are connected and fed from the bus for which the software calculates the summary. The generated report shows the individual loads connected to the bus as well as all the feeders that feed the downstream buses and their loads.

💡 **Tip**

- Note that circuits have a property called **CircuitMode** with values **Connected** and **Disconnected**. If you set this property to **Disconnected**, the drill down algorithm will stop the calculation and the software will not roll up the loads connected to this feeder.

The output of the All Feeder Load Summary report depends on the settings that you make in the Options Manager or on the **Reports** tab of the **Preferences** dialog box. There are two major factors that influence the results of the report:

- The inclusion of coupled buses in the All Feeder Load Summary report.

- The calculation method for the All Feeder Load Summary report.

The following table shows how the inclusion of coupled buses affects the output of the All Feeder Load Summary report. These settings define how the software handles the buses that are coupled with the selected bus and the downstream buses.

Preference Setting	Result
Do not include coupled buses	The software disregards any existing tied bus on the selected PDB. The software also disregards all the buses that are not powered directly from a feeder in the power path. The software generates a summary that is based on all the loads that are connected directly to the current bus and the loads that are connected to the downstream buses that are connected to the current bus. The software excludes all the loads on the redundant or coupled buses as well as all the transformers and converting equipment.
Include coupled buses connected by couplers only	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "coupler circuits". Therefore, all the loads on tied buses are also included in the calculation if the tied bus is connected to the selected bus using a coupler.
Include coupled buses connected by couplers and risers	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "riser circuits". This is the most inclusive option that allows you to include in the calculation not only the loads on tied buses connected by couplers but also those that are connected by riser circuits.
Account for converting equipment power losses	Instructs the software to account for power losses when using converting equipment.

The second major factor that affects the results of the report is the calculation method for the All Feeder Load Summary report.

Selecting the **Use each load coincidence factor** option instructs the software to look for the load operating mode (continuous, intermittent, spare, or standby). The software multiplies the consumed kW and kVAr by the respective coincidence factor (X, Y, Z, ZZ) and uses these new "factored" numbers in the summary. The software calculates two sets of load summaries - the total load for **normal** and **peak** operations.

The calculated total load for **normal** operation is a summary of all the loads whose operating mode is continuous and intermittent. The summary does not include any standby or spare loads. The software calculates this summary based on the following equation:

Sigma load kW = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number]
Sigma load kVAr = Sigma of the [each of the continuous loads kVAr multiplied by their individual X coincidence number + each of the intermittent loads kVAr multiplied by their individual Y coincidence number]
Sigma kVA = Sqrt [Sigma load kW+ Sigma load kVAr]
Average load power factor = Sigma KW/Sigma kVA

The calculated total load for **peak** operation is a summary of all the loads whose operating mode is continuous, intermittent, standby, or spare. The software calculates this summary based on the following equation:

Sigma load kW = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number + each of the standby loads KW multiplied by their individual Z coincidence number + each of the spare loads kW multiplied by their individual ZZ coincidence number]
Sigma load kVAr = Sigma of the [each of the continuous loads kW multiplied by their individual X coincidence number + each of the intermittent loads kW multiplied by their individual Y coincidence number + each of the standby loads kW multiplied by their individual Z coincidence number + each of the spare loads kW multiplied by their individual ZZ coincidence number]
Sigma kVA = Sqrt [Sigma load kW + Sigma load kVAR]
Average load power factor = Sigma kW/Sigma kVA

The **second option** setting for calculating the load summary uses average bus coincidence factors instead of the individual coincidence factors of the loads. The only difference is that in this case, instead of multiplying the loads by their individual coincidence factors, the software multiplies all the loads of a specific bus by the bus X, Y, Z, or ZZ. The equation for this option is as follows:

Sigma load kW = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number]
Sigma load kVAr = Sigma of the [all of the continuous loads kVAr multiplied by the bus X coincidence number + all of the intermittent loads kVAr multiplied by the bus Y coincidence number]
Sigma kVA = Sqrt [Sigma load kW + Sigma load kVAr]
Average load power factor = Sigma KW/Sigma kVA

The total load for peak operation in this case also includes continuous, intermittent, standby, and spare loads. The equation for this option is as follows:

Sigma load KW = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number + all of the standby loads kW multiplied by the bus Z coincidence number + all of the spare loads kW multiplied by the bus ZZ coincidence number]

Sigma load kVAr = Sigma of the [all of the continuous loads kW multiplied by the bus X coincidence number + all of the intermittent loads kW multiplied by the bus Y coincidence number + all of the standby loads kW multiplied by the bus Z coincidence number + all of the spare loads kW multiplied by the bus ZZ coincidence number]

Sigma kVA = Sqrt [Sigma load kW+ Sigma load kVAr]

Average load power factor = Sigma kW/Sigma kVA

In addition to the two options mentioned above, the software also allows you to specify a bus for which you can perform a calculation based on the bus rated values instead of performing a roll up calculation. To make such a calculation, in the **Properties** window of the pertinent bus, set the **Drill Down Flag** property to **False**. This setting instructs the software to calculate the load based on the bus rated power values of the selected bus.

At the bottom of the All Feeder Load Summary report, the software displays the calculated total of all the loads of the selected PDB. The software sums the kW and kVAr values of all the buses and also calculates the kVA and the power factor.

Related Topics
- *Generate a Report for Selected Items*, page 388
- *PDB Load Summary by Operating Mode Report*, page 408
- *Shipped Reports*, page 390
- *Working with the Electrical Index: An Overview*, page 44

All Feeder Load Summary (Enhanced) Report

The All Feeder Load Summary (Enhanced) report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Electrical Engineer**. This report is based on either the consumed or rated power of the selected loads.

The All Feeder Load Summary (Enhanced) report provides a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the coincidence factors of the relevant buses or the individual loads, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. The enhanced report sums the loads that are connected to the downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

Once you start generating an enhanced All Feeder Load Summary report (for details, see *Generate an Enhanced All Feeder Load Summary Report*, page 256), the software retrieves the data of all the loads that are connected to the buses that belong to the PDBs or buses that you selected in the **Electrical Index** or the **Electrical Engineer** and starts the bus drill-down calculation. The software retrieves the values for one bus at a time (the summary is per bus within the selected PDB) including the loads that are connected to the downstream loads that are connected and fed from the bus for which the software calculates the summary. The generated report shows the individual loads connected to the bus as well as all the feeders that feed the downstream buses and their loads.

As opposed to the presentation of data in the regular All Feeder Summary report, the enhanced report shows consumed or rated power values, compensated or uncompensated for the coincidence factors of the buses or loads (depending on the preference settings). For details, see *Customize All Feeder Load Summary Report Preferences*, page 116

Furthermore, the enhanced report also shows the kV and kVAR values for individual loads grouped according to the relevant operation mode of the load. The report also includes the converting equipment power losses data as well as circuit totals.

💡 **Tip**

- Note that circuits have a property called **CircuitMode** with values **Connected** and **Disconnected**. If you set this property to **Disconnected**, the drill down algorithm will stop the calculation and the software will not roll up the loads connected to this circuit.

At the bottom of the All Feeder Load Summary report, the software displays the calculated compensated totals of all the loads at normal and peak operations.

Note that you can customize your Enhanced All Feeder Load Summary report.

For instructions, in your **SmartPlant Electrical Help** window, click **Contents** > **Reporting** > **Customizing the Enhanced All Feeder Load Summary Report**.

Related Topics
- *Calculating Bus Load Summaries: An Overview*, page 234
- *Customize All Feeder Load Summary Report Preferences*, page 116
- *Generate an Enhanced All Feeder Load Summary Report*, page 256
- *Shipped Reports*, page 390

PDB Load Summary by Operating Mode Report

The PDB Load Summary by Operating Mode report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Tabular Editor**. The PDB Load Summary by Operating Mode report is intended to complement the All Feeder Load Summary report, which calculates the running and peak loads according the coincidence factor of each load or according to the bus coincidence factor. The PDB Load Summary by Operating Mode report satisfies the need to calculate a load summary according to special operating modes, such as emergency or life support systems for offshore platforms. The software provides ten operating mode factors (duty and load factors) for each load or bus.

Apart from one major function, the PDB Load Summary by Operating Mode report has the same purpose as the PDB Load Summary report which is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual duty and load factors, calculate the total consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver under ten different operating modes such as emergency or life support. While the PDB Load Summary report calculates **only** the total direct load connected to a bus, the PDB Load Summary by Operating Mode report **also** sums the loads that are connected to downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

Once you start generating an All Feeder Load Summary by Operating Mode report (for details, see *Generate a Report for Selected Items*, page 388), the software starts retrieving the data of all the loads that are connected to the buses that belong to the PDB that you selected in the **Electrical Index**. The software retrieves the values for one bus at a time (the summary is per bus within the selected PDB) including the loads that are connected to the downstream loads that are connected and fed from the bus for which the software calculates the summary. The generated report shows the individual loads connected to the bus as well as all the feeders that feed the downstream buses and their loads.

💡 **Tip**

- Note that circuits have a property called **CircuitMode** with values **Connected** and **Disconnected**. If you set this property to **Disconnected**, the drill down algorithm will stop the calculation there and will not roll up the loads connected to this feeder.

The output of the All Feeder Load Summary by Operating Mode report depends on the settings that you make in the Options Manager or on the **Reports** tab of the **Preferences** dialog box. There are two major factors that influence the results of the report:

- The inclusion of coupled buses in the All Feeder Load Summary by Operating Mode report.

- The calculation method for the All Feeder Load Summary by Operating Mode report.

The following table shows how the inclusion of coupled buses affects the output of the All Feeder Load Summary by Operating Mode report. These settings define how the software handles the buses that are coupled with the selected bus and the downstream buses.

Preference Setting	Result
Do not include coupled buses	The software disregards any existing tied bus on the selected PDB. The software also disregards all the buses that are not powered directly from a feeder in the power path. The software generates a summary that is based on all the loads that are connected directly to the current bus and the loads that are connected to the downstream buses that are connected to the current bus. The software excludes all the loads on the redundant or coupled buses as well as all the transformers and converting equipment.
Include coupled buses connected by couplers only	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "coupler circuits". Therefore, all the loads on tied buses are also included in the calculation if the tied bus is connected to the selected bus using a coupler.
Include coupled buses connected by couplers and risers	The software includes all the loads connected to all the buses encountered in the power flow path including those buses that are connected by "riser circuits". This is the most inclusive option that allows you to include in the calculation not only the loads on tied buses connected by couplers but also those that are connected by riser circuits.

The second major factor that affects the results of the report is the calculation method for the All Feeder Load Summary by Operating Mode report.

Selecting the **Use each load coincidence factor** option instructs the software to use the load and duty factors that are defined for each consumer. The software uses these new "factored" numbers in the summary. The software calculates ten sets of load summaries, each set for a different mode.

Each operating mode (for LoadN) is calculated according to the following equations:

$$ActivePower@ModeX = \frac{BrakePower \times ModeXLF \times ModeXDF}{Efficiency@BrakePower}$$

$$ApparentPower@ModeX = \frac{BrakePower \times ModeXLF \times ModeXDF}{Efficiency@BrakePower \times PF@BrakePower}$$

$$ReactivePower@Modex = \sqrt{(ApparentPower@ModeX)^2 - (ActivePower@ModeX)^2}$$

Where X = Mode designator.

The **second option** setting for calculating the load summary uses average bus load and duty factors instead of the individual load and duty factors of the loads. The only difference is that in this case, instead of multiplying the loads by their individual load and duty factors, the software multiplies all the loads of a specific bus by the bus ModeX load and duty factors. The equations for this option are as follows:

$$ActivePower@ModeX = \frac{BrakePower \times BusModeXLF \times BusModeXDF}{Efficiency@BrakePower}$$

$$ApparentPower@ModeX = \frac{BrakePower \times BusModeXLF \times BusModeXDF}{Efficiency@BrakePower \times PF@BrakePower}$$

$$ReactivePower@Modex = \sqrt{(ApparentPower@ModeX)^2 - (ActivePower@ModeX)^2}$$

Where X = Mode designator.

In addition to the two options mentioned above, the software also allows you to specify a bus for which you can perform a calculation based on the bus rated values instead of performing a roll up calculation. To make such a calculation, in the **Properties** window of the pertinent bus, set the **Drill Down Flag** property to **False**. This setting instructs the software to calculate the load based on the bus rated power values of the selected bus.

At the bottom of the All Feeder Load Summary by Operating Mode report, the software displays the calculated total of all the loads of the selected PDB. The software sums the kW and kVAr values of all the buses and also calculates the kVA and the power factor.

Related Topics
- *All Feeder Load Summary Report*, page 391
- *Generate a Report for Selected Items*, page 388
- *Shipped Reports*, page 390
- *Working with the Electrical Index: An Overview*, page 44

Registering Reports: An Overview

When you create an Excel report in SmartPlant Electrical, you can enable revisions for the report by registering it. If you create a report for an item type where the data can be published, for example, instruments, cabinets or signal runs, you also need to register the report and enable publishing. When you register a report, it appears under **Documents** > **Registered Reports** in the **Electrical Index**. Note that you can associate an electrical item with a registered report. Also, you can compare the current data with the data in an archived revision of a simple registered report.

Related Topics

- *Associate an Electrical Item with a Registered Report*, page 414
- *Include Revision Data in Registered Reports*, page 413
- *Manage Global Revisions*, page 375
- *Register a Report*, page 413

Register a Report

1. Click **Actions** > **Register Report**.

2. On the **Register Report Common Properties** dialog box, beside the **Report template** box, click the ellipsis button ⬜ and select a report file.

   💡 **Tip**

   - You can create a custom report or select one of the reports that is shipped with SmartPlant Electrical. For the full list of shipped reports, see *Shipped Reports*, page 390.

3. If you want to filter the data that appears in the report, beside **Filter**, click the ellipsis button ⬜ and select a filter that is suitable for the report template. For details of how to create or modify a filter, see Filter Manager Online Help.

4. Select a value from the **Document main item type** list if needed.

   💡 **Tip**

   - This value is for information purposes only.

5. Type a name for the document, and if required, a description.

6. To use the report for publishing data to an XML file, select **Enable document for publishing**.

✎ **Notes**

- A report that you register can display revision numbers after you archive the report.

- The registered report with its filter defines the exact scope or subset of data to be published. You can view the layout of this report and the properties that are included in the report when you open the report; however, the actual data (properties) to be published with each item is defined in the code.

Related Topics
- *Associate an Electrical Item with a Registered Report*, page 414
- *Include Revision Data in Registered Reports*, page 413
- *Manage Global Revisions*, page 375
- *Register Report Common Properties Dialog Box*, page 759
- *Registering Reports: An Overview*, page 409
- *Working with Internal Documents Common Tasks*, page 373
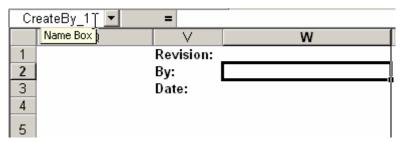
Include Revision Data in Registered Reports

1. Click **Reports** > **Edit**.

2. On the **Edit Report Template** dialog box, select a registered report template.

3. Click **Open**.

4. In the Excel report template, select the cells for the revision headers and type the header text in the selected cells.

   💡 **Tip**

   - If you want the revision information to appear at the top of the report, select the cells in the Header section of the template.

5. Select another cell to contain the revision data.

6. In the **Name Box**, type the appropriate name for mapping the desired revision data, for example:

7. Press **Enter**.

8. After adding all the desired revision data, save the Excel template.

Notes

- The following revision mapping names are available:

- LastRevision (same as RevisionNumber_1)

- RevisionNumber_1

- RevisionDate_1

- CreateBy_1

- CheckBy_1

- ApproveBy_1

- RevisionDescription_1

- You can include as many revision levels as you need. You specify the revision number by adding an underscore (_) followed by a number to the mapping name. The lowest number that follows the underscore retrieves the latest revision. For example, if you want to specify the data indicated for 4 revision numbers in the document in descending sequence, you need to include the mapping names (one name per cell) in the following order:

-

Revision	By	Date	
RevisionNumber_1		CreateBy_1	RevisionDate_1
RevisionNumber_2		CreateBy_2	RevisionDate_2
RevisionNumber_3		CreateBy_3	RevisionDate_3
RevisionNumber_4		CreateBy_4	RevisionDate_4

Related Topics

Associate an Electrical Item with a Registered Report

1. Define a document reference for the registered report that you want to associate. For details, see *Define an External Document Reference*, page 380.

2. In the **Electrical Index**, select an electrical item.

   💡 **Tip**

   - You can also associate multiple electrical items with the same registered report. In the **Electrical Index**, click a folder that contains electrical items and highlight in the list view pane the items that you require.

3. Click **Actions** > **Associate Document**.

4. On the **Associate Documents** dialog box, under **Filter by**, under **Document category**, select **Electrical Documents**. This selection determines the available options in the **Document type** list. You can define and customize document categories in the Data Dictionary Manager.

5. From the **Document type** list, select **System Reports**.

6. Click **Add**.

7. On the **Find** dialog box, click **Find Now** to display the available documents.

   💡 **Tip**

   - The **Find** dialog box finds only those documents that belong to the document category and type that you select on the **Associate Documents** dialog box.

8. Select each document that you want to associate with the selected item and then click **Apply**.

   💡 **Tip**

   - To associate documents belonging to another category or type, return to the **Associate Documents** dialog box, select the desired document category and type, click **Associate** again, and repeat the succeeding steps in this procedure.

Related Topics
   - *Define an External Document Reference*, page 380
   - *Registering Reports: An Overview*, page 409
   - *Working with Documents: An Overview*, page 372
   - *Working with External Documents Common Tasks*, page 378

Creating Report Templates: An Overview

Although the software is delivered with several default report templates, such as Blank, Electrical Load List, PDB Schedule, you may need to create your own custom report templates or modify the delivered templates in order to gather and display the specific information you want. So the software allows you to define your own templates in several different formats and including all manner of information from the design database.

Templates can be formatted in three different ways:

- Tabular
- Fixed
- Composite

The delivered report templates are all tabular format reports. Fixed format templates allow the greatest amount of freedom in formatting your report, and composite report templates, as the name suggests, combine tabular and fixed formatting.

Not only can you completely control the format of your report, but you can control the content of the template also. Each report is based on a unique item type, and the properties that are associated with that item type are readily available to include in your report. But any item that is related in any way to the basic item type of your report makes its properties available to include in the definition of your template, too. For instance, the properties of internal circuit components can be used in a report based on circuits because internal circuit components are related to the circuits that they are part of.

Related Topics

Defining Report Templates Common Tasks

The following tasks are frequently used when you create or modify a report template. For a complete list of tasks, click the appropriate chapter on the **Contents** tab and then click the procedure that you want.

Display the Properties of a Report Template

You can inspect some of the properties that are currently in force for your template, such as the report item type and the report format. For more information, see *Display the Properties of a Report Template*, page 429.

Create a New Blank Report Template

You can define a new report template which is not based on a pre-existing template. For more information, see *Create a New Blank Report Template*, page 418.

Create a Report Template Based on an Existing Template

You can define a new report template by using another template as a source. For more information, see *Create a Report Template Based on an Existing Template*, page 419.

Edit a Report Template

You can edit some of the properties of a report template that is already defined. For more information, see *Edit a Report Template*, page 431.

Define the Contents of Your Report Template

You can edit your report template in Microsoft Excel and specify what item properties are available for you to report on. For more information, see *Define the Contents of Your Report Template*, page 428.

Define the Layout of a Tabular Format Report Template

You can edit your tabular format report template in Microsoft Excel and specify how the labels and data are displayed. For more information, see *Define the Layout of a Tabular Format Report Template*, page 422.

Define the Layout of a Fixed Format Report Template

You can edit your fixed format report template in Microsoft Excel and specify how the labels and data are displayed. For more information, see *Define the Layout of a Fixed Format Report Template*, page 425.

Define the Layout of a Composite Format Report Template

You can edit your composite format report template in Microsoft Excel and specify how the labels and data are displayed. For more information, see *Define the Layout of a Composite Format Report Template*, page 426.

Delete a Report Template

You can delete a report template. For more information, see *Delete a Report Template*, page 431.

📝 **Note**

- The location where plant-level report templates are stored is specified in Options Manager.

Create a New Blank Report Template

1. Click **Reports > New**.

2. On the **New Report Template** dialog box select **Blank** for the **Source template**.

   💡 **Tips**

   - You can create a new report template based on an existing template by specifying a source template also.

     Create a Report Template Based on an Existing Template, page 419

   - Since delivered reports are all in tabular format, you must choose a **Blank** template if you want a fixed or composite format report template. You can use **Blank** for a tabular format report template, too.

3. Type a meaningful name to describe the report template you are creating in the **Name** field. This is the name of your Microsoft Excel workbook. An .xls extension is automatically appended to the name when you save the template.

4. Select an **Item type** to base your report on.

5. Choose template format from **Report type**. These options are only available when you choose **Blank** for **Source template**.

6. You can type a description for the template in the **Description** field.

7. Select **Add to plant reports** if you want this template to be available at the plant level.

   💡 **Tip**

   - You must have the appropriate permissions, granted in SmartPlant Engineering Manager, to create plant report templates.

8. Click **OK** to create the template.

9. When the software gives you the option to edit your new template in Excel, click **OK** and define the layout and contents of the template.

    Define the Contents of Your Report Template, page 428

    Define the Layout of a Tabular Format Report Template, page 422

    Define the Layout of a Fixed Format Report Template, page 425

    Define the Layout of a Composite Format Report Template, page 426

Related Topics
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Edit a Report Template*, page 431
- *Using the Composite Format Report*, page 425
- *Using the Fixed Format Report*, page 423

Create a Report Template Based on an Existing Template

1. Click **Reports > New**.

2. Select an existing template from the **Source template** list on the **New Report Template** dialog box.

3. Type a name for the new report template in the **Name** field.

4. Type a description for the new report template in the **Description** field.

5. Click **OK**.

6. The software gives you the option to edit your template. Click **OK** to open Excel and define the contents and layout of your report.

    Define the Contents of Your Report Template, page 428

    Define the Layout of a Tabular Format Report Template, page 422

    Define the Layout of a Fixed Format Report Template, page 425

    Define the Layout of a Composite Format Report Template, page 426

📝 **Notes**
- Since delivered report templates are all in tabular format, if you create a template using an existing template, the format of your report template will also be tabular.

- Since you are creating a report based on an existing template, some options on the **New Report Template** dialog box are not available. In order to make all options available, you choose **Blank** for your **Source template**.

Defining the Layout of Your Report: An Overview

Define the layout, or appearance, of your report by defining a report template. This template is a Microsoft Excel workbook that contains many cells and worksheets. Properties are mapped to these individual Microsoft Excel cells to define the content and layout of your report. To create a list of report item properties available to include in your template, use the **Define Report Items** dialog box, which is opened when you click **Define** on the **Define Reports Contents** dialog box.

Use the **Options** command on the **SmartPlant Reports** toolbar to designate space for your header and to specify the number of blank lines you want between rows of data on your report. The **Options** command is not available for fixed format report templates because you are free to place headers and data anywhere on your worksheet.

In order to define the layout of the report item properties, you assign properties to particular cells. Select the cell and then select the property from the **Map Properties** menu on the **SmartPlant Reports** toolbar.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Create a Report Template Based on an Existing Template*, page 419
- *Define the Contents of Your Report Template*, page 428
- *Display the Properties of a Report Template*, page 429
- *Edit a Report Template*, page 431
- *Generate a Report*, page 387

Using the Tabular Format Report

All the delivered report templates are tabular format reports. When the report is generated, all properties populate the report by using the same format defined for the first row. In other words, tabular format reports are row-based. The **Options** command on the **SmartPlant Reports** toolbar is important for setting aside space for the header and empty rows between lines in the report because the placement of report item properties is restricted in the tabular format report template.

For example, the report template for a Motor List can appear like this:

Motor Name	Description	Type
#Motor::Name#	#Motor::Description#	#Motor::Type#

When the report is generated, the output appears like this:

Motor Name	Description	Type
101-M-100	Water pump motor	Squirrel cage
101-M-101	Main Feed pump motor	Special

Related Topics
- *Create a New Blank Report Template*, page 418
- *Define the Contents of Your Report Template*, page 428
- *Define the Layout of a Tabular Format Report Template*, page 422
- *Display the Properties of a Report Template*, page 429
- *Edit a Report Template*, page 431
- *Generate a Report*, page 387

Define the Layout of a Tabular Format Report Template

1. Click **Options** on the **SmartPlant Reports** toolbar.

2. On the **Options** dialog box, choose the size of your report header.

   💡 **Tips**

   - The header is part of the tabular format report template where data is not mapped based on rows. That is, you are free to type a label and map report item properties anywhere in the lines you designate for your header.

   - The header is a good place to put information that applies to all the items in your report, such as Unit or Plant Name.

   - Also, you can add graphics in your report header.

   - Be sure to include space in your header for the labels of your columns.

3. Choose the number of empty lines you want between rows in your report. Each row is filled with properties for one item.

   💡 **Tip**

   - In some report templates, it might be useful to designate two rows of data for each report item and then an empty row.

4. Click **OK** to close the **Options** dialog box.

5. Fill in general labels and graphics, if needed, in your header.

6. Fill in column headings in the last row of your header.

7. Select a cell in your report template where you want to map a particular property. This can be either in the body of the report or in the header.

8. Choose the property you want to map to your cell from the **Map Properties** menu. The items on this menu are chosen when you define the contents of your template:

   Define the Contents of Your Report Template, page 428

   💡 **Tips**

   - You only have to map properties for one item. Properties for all the items will follow the format you mapped for the first one.

   - Only one property can be mapped to a given cell, although the same property can be mapped to more than one cell.

9. Repeat steps 7 through 8 until you have mapped all the properties you want to map on this report template.

10. Save the template and quit Excel.

Notes

- Since delivered report templates are all in tabular format, if you create a template based on an existing template, the format of your report template will also be tabular. Use **Blank** as your source template if you want to create a fixed format or composite format report template.

- The location where plant-level report templates are stored is specified in Options Manager.

- When any property value in the header is added, a new Microsoft Excel worksheet is created, and the header is saved on the new sheet because the entire header no longer fits in the space allotted on the first sheet. The number of rows in the report header can be customized using the **SmartPlant Reports** toolbar and selecting **Options** - in the dialog box choose a larger value for **Rows in report header**. For example, if you want to have the loads connected to different PDBs on separate pages, you should map the PDB item tag to the header section. Also, if you want to have a motor list grouped per process unit with motors of unit one on page 1 and motors of unit 2 on another page, include the plant hierarchy property in the header.

- You can customize the number of rows in the report header using the **SmartPlant Reports** toolbar and clicking **Options** on the dialog box. To increase the number of rows, choose a larger value under **Rows in report header**.

Related Topics

Using the Fixed Format Report

The fixed format report creates one Excel worksheet for each item. When defining the report template you only edit the first worksheet. When you generate your report, all fixed format worksheets, one for each item of the report item type, will follow the format of Excel worksheet one. The **Options** button on the **SmartPlant Reports** toolbar is not available for editing a fixed format report template because you are free to place headers and data anywhere on your worksheet.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Create a Report Template Based on an Existing Template*, page 419
- *Define the Layout of a Fixed Format Report Template*, page 425
- *Display the Properties of a Report Template*, page 429
- *Edit a Report Template*, page 431

Define the Layout of a Fixed Format Report Template

1. Choose a cell on your Excel worksheet and type labeling information into it.

   💡 **Tip**

   - A label is not a required feature of a fixed format report template. You can simply map properties to cells without any labeling if you want to.

2. Choose a cell where you want the property associated with the labeling that you typed in step 1 to be displayed.

3. Click **Map Properties** on the **SmartPlant Reports** toolbar, and from the menu select the property that you want to map to the cell you chose in step 2.

   💡 **Tip**

   - The properties displayed on the **Map Properties** menu are specified when you define the contents of your report.

     Define the Contents of Your Report Template, page 428

4. Repeat steps 1 through 3 until you have labeled and mapped all the needed report item properties to the corresponding locations in your template. When you generate a fixed format report, the data for each report item is displayed in its own worksheet, and the layout for each worksheet matches the layout defined for worksheet 1.

Related Topics
- *Using the Composite Format Report*, page 425
- *Using the Fixed Format Report*, page 423
- *Using the Tabular Format Report*, page 421

Using the Composite Format Report

The composite format report template is a combination of fixed and tabular format reports. In your Microsoft Excel workbook the first sheet contains the fixed format report, and the second sheet contains the tabular format report. You choose your report format on the **Report Properties** dialog box.

📝 Note

- Since delivered report templates are all in tabular format, choose Blank for **Source template** if you want to create a composite format template.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Define the Layout of a Composite Format Report Template*, page 426
- *Display the Properties of a Report Template*, page 429
- *Edit a Report Template*, page 431
- *Generate a Report*, page 387

Define the Layout of a Composite Format Report Template

1. The first Excel worksheet of the composite format report template is a fixed format report:

   Define the Layout of a Fixed Format Report Template, page 425

   ### 💡 Tips

   - Each item in the report has its own page. The properties of the first item are displayed on the first worksheet, but the properties for subsequent items in the report are displayed starting with the third sheet because the second worksheet is tabular format.

   - You use the **Options** command on the **SmartPlant Reports** toolbar to designate space for your header and to specify the number of blank lines you want between rows of data on your report. The **Options** command is not available for fixed format report templates because you are free to place headers and data anywhere you want to on your worksheet; however, the **Options** command is available for you to use in the tabular portion of the composite report template.

2. The second Excel worksheet of the composite format report template is a tabular format report:

   Define the Layout of a Tabular Format Report Template, page 422

3. After you have defined the layout for the first and second worksheets in your report template, save the template and quit Excel.

✎ Notes

- Only one property can be mapped to a given cell, although the same property can be mapped to more than one cell.

- When any property value in the header is added, a new Microsoft Excel worksheet is created, and the header is saved on the new sheet because the entire header no longer fits in the space allotted on the first sheet. The number of rows in the report header can be customized using the **SmartPlant Reports** toolbar and selecting **Options** - in the dialog box choose a larger value for **Rows in report header**. This applies to the tabular portion of your composite format report template. For example, if you want to have the loads connected to different PDBs on separate pages, you should map the PDB item tag to the header section. Also, if you want to have a motor list grouped per process unit with motors of unit one on page 1 and motors of unit 2 on another page, include the plant hierarchy property in the header.

- You can customize the number of rows in the report header using the **SmartPlant Reports** toolbar and clicking **Options** on the dialog box. To increase the number of rows, choose a larger value under **Rows in report header**.

Related Topics

- *Creating Report Templates: An Overview*, page 416
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Using the Composite Format Report*, page 425
- *Using the Fixed Format Report*, page 423
- *Using the Tabular Format Report*, page 421

Defining the Contents of Your Report: An Overview

A report definition describes how to collect the properties data and how to format it in a report. Each report item is based on an item type - motor, load, control station, and so forth - and controls how the properties of that item type, or item types related to it, are retrieved from the database.

You can modify the report definition using the **Define Report Contents** dialog box, which is opened when you click **Define** on the **SmartPlant Reports** toolbar.

You define the layout, or appearance, of your report by formatting the item properties and headings in your report template. The template is displayed in a Microsoft Excel workbook that contains many cells and worksheets. Item properties are mapped to the individual cells. Create a list of properties to be available for your report by using the **Define Report Items** dialog box.

To assign a property to a particular cell, select the cell and then select the property from **Map Properties** on the **SmartPlant Reports** toolbar.

Related Topics

Define the Contents of Your Report Template

1. In Microsoft Excel, click **Define** on the **SmartPlant Reports** toolbar.

2. If you want to add a new report item type so that its properties will be available to map onto your report template, select the node in the **Report on** hierarchy under which you want the new item type to appear. Click **New**.

   Tips

   - When you click **New** on the **Define Report Contents** dialog box, you can choose from only those items which are related to the item you selected in the **Report on** hierarchy.

   - The highest node in the **Report on** hierarchy that you can add a new item under is the node that belongs to the report item type that you base your template definition on. You can add items under any nodes subordinate to the main report item type as long as they have items related to them.

   - Using the **New** command skillfully allows you to navigate in all directions in the plant hierarchy and gather properties from throughout the plant database.

3. On the **New Items** dialog box, select the new item you want to add to the hierarchy.

4. You can change the name of the item that is displayed in the hierarchy by typing your choice in the **Name** box.

   Tip

   - If the item you choose has the same name as an item elsewhere in the hierarchy, you must type a different name for it in the **Name** box.

5. Click **Apply**.

6. When you have added all the items you want from this list, click **Close**.

   Tip

   - If you want to add more new items under another node in the hierarchy, choose that node and repeat steps 2 through 5.

7. On the **Define Report Contents** dialog box, select an item whose properties you want to be available for your report template.

8. Click **Define**.

9. On the **Properties** tab of the **Define Report Items** dialog box, choose the properties that you want to be available to map to your report template.

   💡 **Tips**

   - If you want to sort the order in which your items are listed when you generate a report, choose the property or properties to sort on by using the **Sort** tab.

   - If you want to change the filter that finds your report items, use the **Filter** tab. For instance, you may want to change from a filter that displays all pumps to using a filter that displays only active pumps.

10. Click **OK**. The **Define Report Contents** dialog box remains open, and so you can repeat steps 2 through 9 to continue adding more items and specifying their properties.

11. When you have chosen all the properties you want in your report, click **OK** on the **Define Report Contents** dialog box.

   ✎ **Notes**

   - All the properties you have specified to be available to map into your template are now displayed when you click **Map Properties** on the **SmartPlant Reports** toolbar. Now you can use the properties you have specified in the layout of your template.

   - You can add plant and plant group column headers to your templates. Use the following macros to show plant and plant group names. Note that all macro names are case sensitive:

   - Plant.Name

   - Area.Name

   - Unit.Name

   - Unit.<custom property>

   - You can use custom properties to add client information.

   - For project and contractor information, use the following:

   - Project.<PropertName>

   - Contractor.<PropertName>

   Related Topics

Display the Properties of a Report Template

1. Select **Reports > Edit**.

2. Select a report from the **Available reports** list on the **Edit Report Template** dialog box.

3. Click **Properties**.

Related Topics
- *Creating Report Templates: An Overview*, page 416
- *Edit a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

Edit a Report Template

1. Click **Reports > Edit**.

2. On the **Edit Report Template** dialog box, select a report from the **Available reports** list.

3. You can click **Properties** on the **Edit Report Template** dialog box in order to edit template properties on the **Report Properties** dialog box.

   💡 **Tip**

   - Since you are modifying a template that already is defined, changing the **Report type** and **Description** are the only available actions on the **Report Properties** dialog box.

4. Click **Open** on the **Edit Report Template** dialog box to display the report in Microsoft Excel so that you can edit the layout and contents of your report template:

   Define the Layout of a Tabular Format Report Template, page 422

   Define the Layout of a Fixed Format Report Template, page 425

   Define the Layout of a Composite Format Report Template, page 426

   Define the Contents of Your Report Template, page 428

   📝 **Notes**

   - If the **SmartPlant Reports** toolbar is missing when editing report templates in Microsoft Excel, then from the Excel menu bar, select **Tools > Macro > Visual Basic Editor** and follow these steps:

1. Select **View > Immediate Window**.

2. Type Sheet1.Application.CommandBars("SmartPlant Reports").Delete in the **Immediate Window**.

3. Quit Excel, and the toolbar will be displayed the next time a report is edited.

- You must have the correct permissions, granted in SmartPlant Engineering Manager, to edit a report template.

Related Topics
- *Creating Report Templates: An Overview*, page 416
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Generating Reports: An Overview*, page 386

Delete a Report Template

1. Click **Reports > Delete** on the main menu bar.

2. On the **Delete Report Template** dialog box, select a report from the **Available reports** list.

3. Click **OK** to delete the report template.

4. Click **Close**.

✛ Important

- You must have the correct privileges, granted in SmartPlant Engineering Manager, to delete a report template.

Related Topics
- *Edit a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

SmartPlant Electrical KKS Overview

SmartPlant Electrical supports KKS naming conventions for process-related identification of the following item types:

- PDBs
- Circuits
- Loads
- Converting equipment
- Disconnect electrical equipment
- Signal runs
- Panels (control stations, cabinets, junction boxes, local panels)
- Cables

In addition, the software supports point of installation identification for buses and cells.

For each of these item types, the following KKS segment properties are available:

KKS Level Name	Property	Description
Total Plant	Total Plant	Single character (numeric or alphabetic).
System Code	System Code Prefix (optional)	Single digit.
System Classification	Three-character alphabetic key, selected from a standard list, that represents the type of system in use the plant.	
System Numbering	Two-digit number with leading zeros.	
Equipment Unit Code	Equipment Unit Classification	Two-character alphabetic key, selected from a standard list, that represents the type of equipment within the system used for measuring or monitoring the system, or the type of measurement circuit (level, flow, and so forth).
Equipment Unit Numbering	Three-digit number with leading zeros.	

KKS Level Name	Property	Description
Equipment Unit Additional Code	Optional character (alphabetic) used for special cases such as a thermocouple with dual elements.	
Component Code	Component Classification	Two-character alphabetic key, selected from a standard list, that represents the type of component.
Component Numbering	Two-digit number with leading zeros.	
Numbering Element (for cables only)	Tag Sequence Number (non-KKS)	Four-character numeric or alphabetic value representing the unique numbering of a particular cable.

Total Plant

This property is a single character (numeric or alphabetic).

System Code

System Code Prefix (optional) — Single digit.

System Classification — Three-character alphabetic key, selected from a standard list, that represents the type of system in use the plant.

System Numbering — Two-digit number with leading zeros.

Equipment Unit Code

Equipment Unit Classification — Two-character alphabetic key, selected from a standard list, that represents the type of equipment within the system used for measuring or monitoring the system, or the type of measurement circuit (level, flow, and so forth).

Equipment Unit Numbering — Three-digit number with leading zeros.

Equipment Unit Additional Code — Optional character (alphabetic) used for special cases such as a thermocouple with dual elements.

Component Code

Component Classification — Two-character alphabetic key, selected from a standard list, that represents the type of component.

Component Numbering — Two-digit number with leading zeros.

Numbering Element (for cables only)

Tag Sequence Number (non-KKS) — Four-character numeric or alphabetic value representing the unique numbering of a particular cable.

Examples

For a switchgear or PDB, only the System Code segments are required in most cases. Thus a suitable KKS name for a medium voltage PDB would be '1 0BBA01', where the values are designated as follows:

Property	Value	Description
Total Plant	1	
System Code Prefix	0	
System Classification	BBA	BBA MV distribution board
System Numbering	01	
Equipment Unit Classification	(Not used)	
Equipment Unit Numbering	(Not used)	

For a motor, the KKS identification '1 0PAC01 AP001B – M01' could be used, where the values are designated as follows:

Property	Value	Description
Total Plant	1	
System Code Prefix	0	
System Classification	PAC	Circulating (main cooling) water pump system
System Numbering	01	
Equipment Unit Classification	AP	Pump units
Equipment Unit Numbering	001	
Component Classification	-M	Motors
Component Numbering	01	

🖉 Note

- When making electrical associations, KKS naming is propagated between certain items. For a complete list of propagation rules, see *Propagation of KKS Names in SmartPlant Electrical*, page 438.
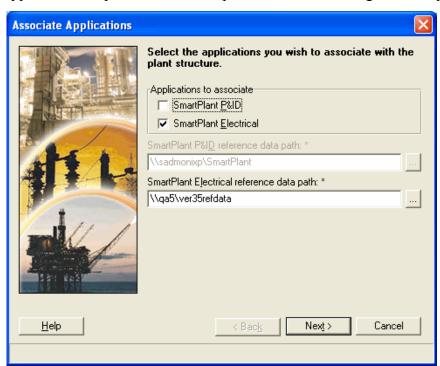
Related Topics
- *SmartPlant Electrical Requirements for Working in KKS Mode*, page 433

SmartPlant Electrical Requirements for Working in KKS Mode

Before you can work in KKS mode in SmartPlant Electrical, you must create a KKS plant in SmartPlant Engineering Manager and in Options Manager, you must select KKS mode and define naming conventions.
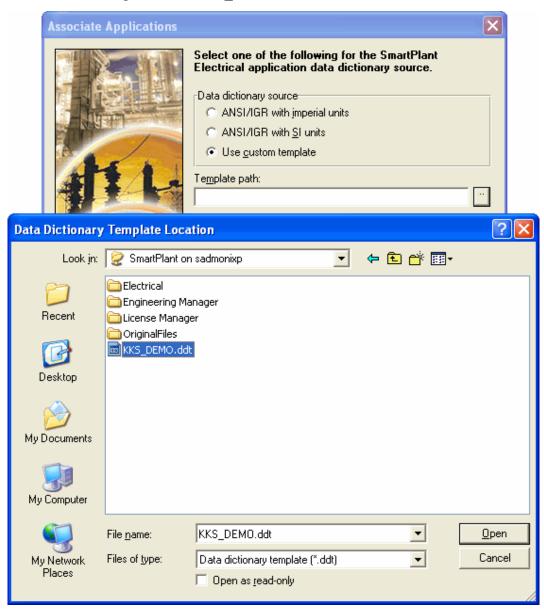
SmartPlant Engineering Manager Requirements

1. Download the SmartPlant Engineering Manager KKS upgrade files. These include the KKS_DEMO.ddt file, which you should place in the **Templates** folder.

2. Create a new plant for use with KKS naming.

3. Expand the tree view for your newly-created plant and right-click the **Applications** folder.

4. On the shortcut menu, click **Associate Applications**.

5. On the first wizard page, under **Applications to associate**, select the **SmartPlant Electrical** check box and in the **SmartPlant Electrical reference data path** box, type the folder path or use the ellipsis button to navigate to the path.

6. Click **Next**.

7. On the second wizard page, under **Data dictionary source**, click **Use custom template**, and in the **Template path** box, type the folder path or use the ellipsis button [...] to navigate to the KKS_DEMO.ddt file.

8. Add roles to the plant as desired.

Options Manager Requirements

1. Click **Options** > **General Settings**.

2. In the **General Settings** window, for the **KKS mode** option, select **Yes**.

   💡 **Tip**
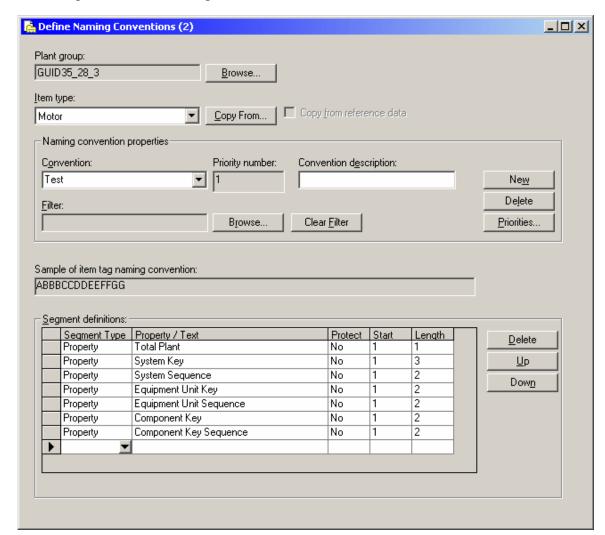
   - Selecting this option activates propagation of KKS names.

3. Click **Options** > **Project-Wide Parameters**.

4. In the **Project-Wide Parameters** window, select the desired plant group and for each item type where you require KKS naming, specify an appropriate default value for the **Component Classification** property.

5. Click **Options** > **Naming Conventions** > **Copy for Plant Group**.

6. Select **Copy from reference data** and select the desired source and target plant groups.

   💡 **Tip**

   - To preview the naming conventions in the *reference plant group* that you copied to your plant, click **Options** > **Naming Conventions** > **View in Reference Data**.

7. Click **Options** > **Naming Conventions** > **Define**.

8. On the **Define Naming Conventions** dialog box, modify the naming conventions for each item type as desired.

   💡 **Tip**

   - The **Define Naming Conventions** dialog box opens with the KKS naming conventions already selected by default. You can delete segment properties that you do not require or add freeform properties in addition to the KKS segments; however, note that in KKS mode, SmartPlant Electrical does not propagate properties of non-KKS segments.

An example of a KKS naming convention for a motor:

✎ Note

- When propagating KKS naming, the software copies the naming from the source item segments to the appropriate target item segments. For this reason, in the naming convention of a target item, you should not delete segments that are likely to receive values by propagation. Non-relevant segments in the target item are left blank. For a complete list of propagation rules, see *Propagation of KKS Names in SmartPlant Electrical*, page 438.

Related Topics
- *SmartPlant Electrical KKS Overview*, page 432
- *Using Naming Conventions*, page 127

Propagation of KKS Names in SmartPlant Electrical

When an item, such as a motor, has KKS naming, and you make an electrical association, for example to an MCC or a control station, the software automatically propagates the naming to the associated item. The software also propagates naming when you create a hierarchy such as PDB > Bus > Feeder in the Electrical Index and then rename the source item. The following table shows how the software handles propagation, along with naming examples.

Target Item Type	Inherited Components of Target Item Name	Possible Source Item Types	Example of Source Item Name
Bus	All populated segments of PDB tag	PDB	
Cell	All populated segments of bus tag + number in group	Bus	
Control station	KKS System and Equipment Unit Codes		
Feeder circuit and internals	Load item tag + KKS segments	All loads	1 0PAC01 AP01B − E01 (heater)
Instrument	KKS System and Equipment Unit Codes		
Space heater	KKS motor naming + heater component code, for example: 1 0PAC01 AP01B − E01	Motor	1 0PAC01 AP01B − M01
Incomer circuit and internals	Upstream power source KKS tag segments	Upstream power source	
Transformer component	KKS System Code of connected item + KKS numeric segments for secondaries, for example: GT002, GT003	Transformer (main item)	GT
I/O signal	KKS System and Equipment Unit Codes + signal application / origin code	Motor, feeder circuit, control station, instrument	

For more details of how propagation works, see *Example of KKS Naming Propagation*, page 440.

Notes

- The software performs propagation according to hard-coded rules. Name segments that are not included in a propagation rule remain blank.

- On connecting a cable side to a load or a panel, the KKS naming is propagated to the cable. When both sides of a cable are connected, the naming comes from the first alphabetic item. On disconnection, the cable retains the name it received by propagation until you reconnect it.

Related Topics
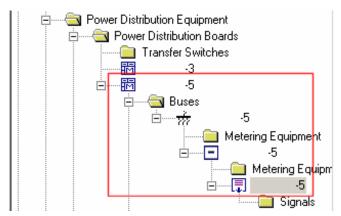
- *SmartPlant Electrical KKS Overview*, page 432

Example of KKS Naming Propagation

This example shows how the software propagates naming when you associate a motor with a control station to a PDB > Bus > Cell > Feeder hierarchy.

Naming an item in an already created hierarchy
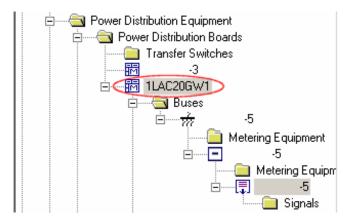
1. In the Electrical Index, create a hierarchy such as PDB > Bus > Cell > Feeder:

2. Rename the source item (PDB):
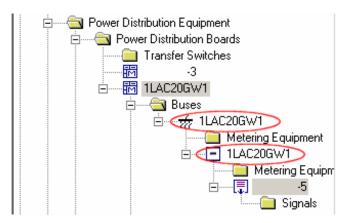
3. View the result in the Electrical Index:

4. Click **Refresh** [icon] to update the display.

    The software propagates the PDB naming to the bus and the cell as shown:
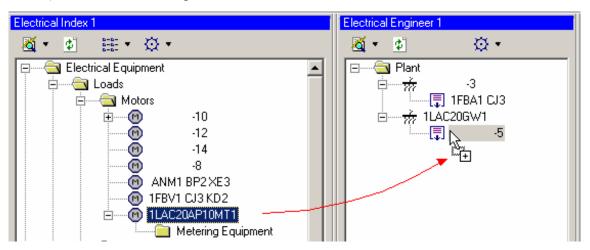
    Note that the propagation does not continue to the Feeder circuit.

Associating one item with another one by dragging
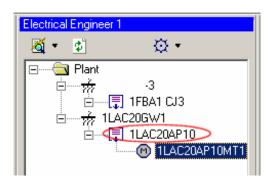
1. Create a new motor and assign it the following name:

2. Drag the motor from the Electrical Index and drop it on the target item (feeder circuit) in the Electrical Engineer:
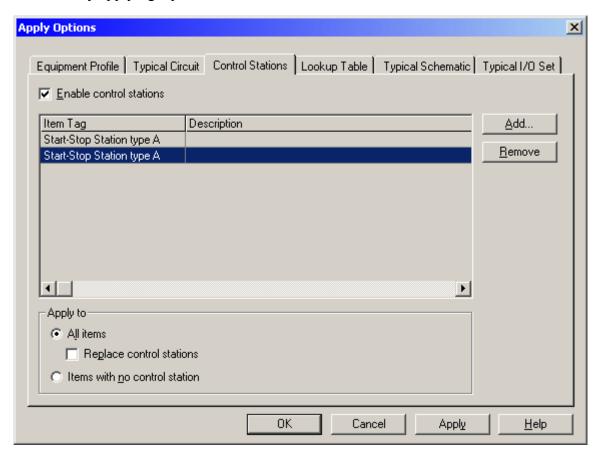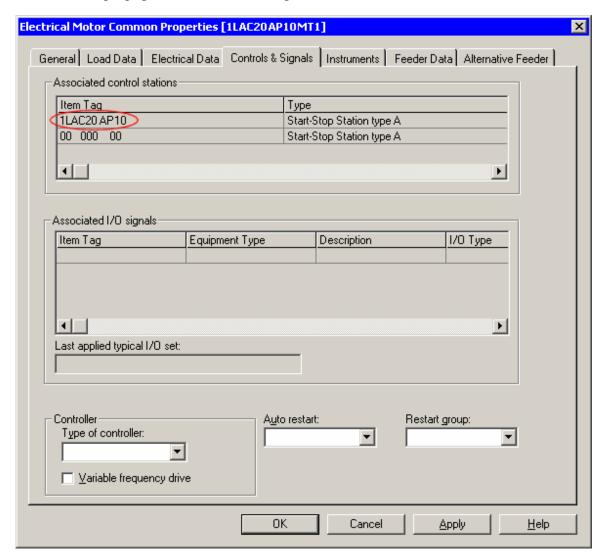
The software propagates the motor naming to the feeder circuit as shown:

Associating one or more items with another one by applying a profile

This example shows how naming is propagated when you add control stations to the motor by applying a profile:

The software propagates the motor naming to the **first** control station as shown:

✎ Note

- Adjust the naming of the target item tags manually if desired.

Related Topics
- *Rename an Item in KKS Mode*, page 446
- *SmartPlant Electrical KKS Overview*, page 432

Scope of KKS Issues for the Current SmartPlant Electrical Version

The current version of SmartPlant Electrical supports KKS naming conventions for process-related identification of most electrical equipment items.

Out of Scope Issues

The following issues are not supported in the current development:

- Operations in an integrated environment

- Configuration rules for KKS establishments

- Modification of the KKS classification keys

- Copying KKS data from one plant group to another

- Grouping of items hierarchically by a common physical system classification or equipment unit

- Point of installation identification, other than for buses and cells

- Location and connection identification

- Signal identification (signal area, signal applications, and gated signals)

- KKS naming conventions for cableways, cableway segments, and cable drums

Notes

- The software does not automatically increment numbering properties of KKS segments.

- When creating multiple items, the names of all the items, other than the first one, appear in the interface as zeros, for example: '00000', even though KKS naming is actually applied to them. In such cases, the user needs to rename the individual items by changing the appropriate numbering segment, after which the software displays the item names correctly.

- The software does not reset cable names when you disconnect one of the cable sides from an associated item that originally propagated its naming to the cable.

- Moving an item to a different level in the Electrical Engineer or changing item properties does not affect the item name.

- If you change the naming convention, it only affects new items. Existing items retain their names according to the previous naming convention.

- The software does not make any validations between the item type and the KKS code.

- The software does not validate imported data.

Related Topics
- *SmartPlant Electrical KKS Overview*, page 432
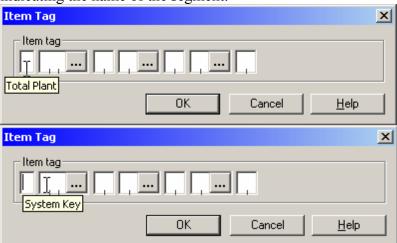
Rename an Item in KKS Mode

1. Select an item and do one of the following:

   - On the item's **Common Properties** dialog box, beside **Item tag**, click the ellipsis button 	.

   - Right-click the item, and on the shortcut menu, click **Rename**.

2. Type appropriate values in each segment of the item tag.

   💡 **Tips**

   - Move the mouse pointer over a segment to display a Tool Tip indicating the name of the segment.

   - Click the ellipsis button 	 beside a Classification segment to display the appropriate **Classification** dialog box, from which you can select suitable values. If you have already typed part of the key prior to clicking the ellipsis button, the **Classification** dialog box opens at the appropriate place in the list.

   - When a numeric segment accepts more than one digit, if you enter fewer digits than the maximum, the software automatically adds leading zeros to the value you typed. You cannot remove the leading zeros from the segment.

The following is an example of a KKS tag name for a motor:

KKS Classification Dialog Box

These dialog boxes allow you to select suitable classification keys when defining names of items that use KKS naming conventions. The specific classification name depends on the segment that you selected in the **Item Tag** dialog box.

Find key — Allows you to type a 2- or 3-character code to drill down to an item in the **Select key** list.

Select key — Displays the set of available keys in a 2- or 3-level hierarchy. Click ⊞ to expand the current portion of the hierarchy further and drill down until the full key appears in the **Find key** field. You can hold the mouse pointer over longer values to display a Tool Tip showing the full name. To select the key for the name segment, double-click the key or click the **OK** command. Note that the **OK** command is only enabled when you select the appropriate level of the hierarchy.

SmartPlant Instrumentation Interface: An Overview

The SmartPlant Instrumentation Interface allows you to exchange data between the two applications using the following document types:

Power elements - You initially define power supply requirements in SmartPlant Instrumentation for tag numbers and for any panel type except for junction boxes and device panels. On publishing the data from SmartPlant Instrumentation, SmartPlant Electrical uses the published power supply data to create instruments from SmartPlant Instrumentation tag number data and cabinets from panel data in SmartPlant Instrumentation. You can then publish the data and retrieve it back to SmartPlant Instrumentation with SmartPlant Electrical PDB information.

Signals - Signal data originates in SmartPlant Electrical. You can define signal data for generators, battery banks, converting equipment, loads, control stations, any type of circuit, and disconnect equipment. On retrieving all SmartPlant Electrical signals, SmartPlant Instrumentation creates special electrical tags, for which you can perform wiring and I/O control system tag assignments. After the wiring is complete, you can publish the data back to SmartPlant Electrical for further processing, and use it to display PLC or other host I/O data in SmartPlant Electrical schematics.

Notes

- The documents published by the two applications do not contain the same properties. For example, instrument and cabinet data includes power supply parameters such as rated voltage, full load current, and so forth, which originate in SmartPlant Instrumentation. However, SmartPlant Electrical only publishes PDB/Cell/Circuit information. Similarly, SmartPlant Instrumentation publishes host data for signals that originate in SmartPlant Electrical, but SmartPlant Electrical does not publish the host data back to SmartPlant Instrumentation. For this reason, if, in SmartPlant Electrical, you change data values that originated from SmartPlant Instrumentation, you should update those values accordingly in SmartPlant Instrumentation, and vice versa.

- If you publish signals in SmartPlant Electrical for circuits that feed items other than loads, converting equipment, generators, battery banks, instruments, or cabinets, then on retrieving the items in SmartPlant Instrumentation and running the tasks, SmartPlant Instrumentation does not display the associated items.

- Whenever you run tasks in the interface that create or update items in SmartPlant Electrical, the software does not validate the data; however, you can select the items and perform batch data validation (for details, see *Display Rule Inconsistency Statistics*, page 122).

Prerequisites for Working with the SmartPlant Instrumentation Interface

Before you can work with the SmartPlant Instrumentation Interface, do the following:

1. Install the latest version of the MSXML Parser prior to installing SmartPlant Electrical.

    💡 **Tip**

    *   During SmartPlant Electrical installation, the software prompts you to install MSXML Parser if it is not already installed on your computer.

2. Install the Intergraph Schema Component from the SmartPlant Electrical CD Browser (click **Add-In Software**, and then click **Schema Component Installation**).

    💡 **Tip**

    *   On the **Select Optional Features** page, make sure that you clear the **Schema Editor** check box.

3. Install the Intergraph SmartPlant Client from the SmartPlant Electrical CD Browser (click **Add-In Software**, and then click **SmartPlant Client Installation**).

    💡 **Tip**

    *   On the **Select Features** page, make sure that you clear the **SmartPlant Automatic Retrieval** check box.

4. In SmartPlant Engineering Manager, set access rights for publish and retrieve to **Full**.

5. In SmartPlant Format Manager, set the accuracy or units of the following properties to be the same as the corresponding properties in SmartPlant Instrumentation to ensure that there are no mismatches when retrieving data into SmartPlant Electrical:

6.

Property	Units or Accuracy Value
Instrument and cabinet rated power	kW
Rated active, reactive, and apparent loads	kW, kVAr, kVA
Full load current and starting current	A
Coincidence factors x, y, z, zz and power factors at 100%	Format = g3

7. In SmartPlant Engineering Manager, under **Hierarchy Templates**, set up a plant hierarchy with an identical plant structure and with identical plant group names to the plant hierarchy items in SmartPlant Instrumentation for which you are going to publish and retrieve data.

8. In SmartPlant Electrical Options Manager, under **Item Types**, select the following values for the properties indicated:

9.

Item Type	Plant group type	Item type uniqueness level	Is unique (check box)
Cabinet	Plant (or equivalent highest level plant group)	Plant	Select
Instrument	Unit (or equivalent lowest level plant group)	Unit	Select
SignalRun	Unit (or equivalent lowest level plant group)	Unit	Select

10. In the SmartPlant Electrical program, publish the plant group data and structure (click **SmartPlant** > **Publish Plant Groups**). This operation creates a file, Plant Groups.xml in the folder of the current plant. SmartPlant Instrumentation uses the data in this file to map and register the SmartPlant Electrical plant groups in the new SmartPlant Instrumentation domain.

Note

- Before working with the interface, ensure that the appropriate prerequisites are also met on the SmartPlant Instrumentation side. For details, refer to the SmartPlant Instrumentation Online Help topic: *Prerequisites for Working with the SmartPlant Instrumentation Interface*.

Related Topics
- *Mapping Plant Groups to SmartPlant Instrumentation*, page 450
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Mapping Plant Groups to SmartPlant Instrumentation

In SmartPlant Electrical Options Manager, you can determine for each item type the plant hierarchy level to which that item type belongs, and at which level the item tags are defined as being unique. In cases where uniqueness applies at a lower level than the entire plant, such as per unit (or equivalent lower level plant group), it is possible for items with identical tags to exist in different units. For example, you can have two motors in two different units, each of which has an identical signal, JSM-100. If you then publish data for these signals for retrieval to the same target unit, SmartPlant Instrumentation will not be able to distinguish between them and the data does not get transferred correctly. Moreover, on transferring data back to SmartPlant Electrical, the software cannot determine to which SmartPlant Electrical unit each set of data belongs.

To ensure that data published from SmartPlant Electrical is defined uniquely in SmartPlant Instrumentation, you must define unique mapping between the plant groups in the two applications. In this way, SmartPlant Instrumentation recognizes each SmartPlant Electrical item as being unique.

Related Topics
- *Map Plant Groups to SmartPlant Instrumentation*, page 452
- *Prerequisites for Working with the SmartPlant Instrumentation Interface*, page 449
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Map Plant Groups to SmartPlant Instrumentation

1. In SmartPlant Engineering Manager, under Hierarchy Templates, set up a plant hierarchy with an identical structure to the one in SmartPlant Instrumentation.

2. Create plant groups for SmartPlant Electrical with identical names to the plant hierarchy items in SmartPlant Instrumentation for which you are going to publish and retrieve data.

3. In SmartPlant Electrical, click **SmartPlant** > **Publish Plant Groups**.

4. On the **Publish to File** dialog box, specify the folder in which to save the generated .xml files by typing the path in the **Folder to save XML files** box.

5. In SmartPlant Instrumentation, log in to the default plant of the desired domain and retrieve the published .xml file containing the SmartPlant Electrical plant group data and structure.

6. Run the generated tasks from the SmartPlant Instrumentation To Do List to create the target plant hierarchy structure.

Notes

- After you have defined the mapping between the source and target units (or equivalent lowest level plant group) and you have transferred data to SmartPlant Instrumentation, you should not change the target unit mapping definitions, because the data cannot be moved in SmartPlant Instrumentation in certain cases (for example where no naming convention is defined for the target unit).

- You can create additional units in the SmartPlant Electrical plant group hierarchy if needed. In this case, you must run the **Publish Plant Groups** command again to create those units in SmartPlant Instrumentation and map them to the SmartPlant Electrical units.

Related Topics

Creating Signals

Signals are items that allow you to transfer I/O data between SmartPlant Electrical and SmartPlant Instrumentation and include properties pertaining to both applications. After creating the signals, you publish the signal data to SmartPlant Instrumentation. SmartPlant Instrumentation uses signals to create special tag numbers that are associated with SmartPlant Electrical and may be wired to PLC and DCS panels as regular I/O signals. After the signals are wired and associated with I/O channels in SmartPlant Instrumentation, you can retrieve the I/O information back to SmartPlant Electrical and use it to add as item types with macros in schematics. These item types display PLC or other host I/O data such as DCS data in the schematics.

You can create signals for all load items, converting equipment, disconnect electrical equipment, control stations, and power distribution board circuits.

Electrical signals include properties that you can edit in SmartPlant Electrical such as **Item Tag**, **Tag Prefix**, **Tag Suffix**, **Description**, **Design By**, **Sequence in Group**, and **System I/O Type**. Other properties belong to SmartPlant Instrumentation and are read-only, for example, **Host Channel Address**, **Reference Loop Number**.

Note

- You can add or delete signals provided you have access rights for Electrical Equipment, regardless of the item type to which you add the signal.

Related Topics
- *Create a Signal*, page 454
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Create a Signal

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.

2. Do one of the following:

   - Navigate to one of the **Loads** folders and select an item.

   - Navigate to one of the **Converting Equipment** folders and select an item.

   - Expand the hierarchy **Power Distribution Equipment** > **Disconnect Electrical Equipment**, then navigate to one of the folders and select an item.

   - Expand the hierarchy **Power Distribution Equipment** > **Power Distribution Boards** and select a circuit.

   - Expand the hierarchy **Wiring Equipment** > **Panels** > **Control Stations** and select a control station.

3. Right-click the selected item, and then click **New Signal**.

4. Edit the signal properties in the **Properties** window as required.

   ### 💡 Tip

   - Some of the properties that you can edit include **Item Tag**, **Tag Prefix**, **Tag Suffix**, **Description**, **Sequence in Group**, and **System I/O Type**; there are other properties that belong to SmartPlant Instrumentation and are read-only.

   ### 📝 Note

   - You can also create signals for an item by dragging a typical I/O set in the **Reference Data Explorer** onto the item in the **Electrical Index**.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Creating Signals*, page 453
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Retrieve Data from the SmartPlant Instrumentation Interface

Follow these steps to retrieve data from SmartPlant Instrumentation:

1. Open SmartPlant Electrical.

2. Click **SmartPlant** > **Retrieve**.

3. On the **Retrieve Document** dialog box, type the path to the folder where the source .xml files containing the published data are located, or click **Browse** to navigate to the folder.

4. From the **Document** list, select a document to retrieve.

   💡 **Tip**

   - The software automatically retrieves the appropriate .xml files associated with the document that you select.

5. Click **SmartPlant** > **To Do List**.

6. On the **To Do List** dialog box, highlight one or more tasks (hold down **Ctrl** for multiple selection) and on the toolbar, click one of the following icons:

   - ▶ - Runs the selected tasks.

   - ▬ - Defers the selected tasks.

   - ✖ - Deletes the selected tasks.

   - 🖹 - Displays the properties of the selected tasks.

   📝 **Notes**

   - The software retrieves data into the appropriate plant groups at various levels of the plant hierarchy, regardless of the plant group that you are currently logged in to.

   - When you run a task that creates a new cabinet or instrument, the software changes the value of the **Override FLA calculation** property to **Manual** to ensure that there is no inconsistency between SmartPlant Electrical and SmartPlant Instrumentation for the values of the **Full load current** property.

   - If you view the properties of a **Create** task, the destination item tag is the same as the source item tag until you run the task. After you run the task, the destination item tag receives the appropriate SmartPlant Electrical designation.

- It is possible for SmartPlant Instrumentation users to delete items that originated from SmartPlant Electrical, such as signals. In this case, when data is next published from SmartPlant Instrumentation and retrieved into SmartPlant Electrical, the software creates **Delete** tasks for those items on the **To Do List** dialog box. If you do not want those items to be deleted from the SmartPlant Electrical database, you should click ✕ to remove the task from the **To Do List** dialog box.

Related Topics
- *Creating Signals*, page 453
- *Prerequisites for Working with the SmartPlant Instrumentation Interface*, page 449
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Publish Data from SmartPlant Electrical

Follow these steps to publish data for SmartPlant Instrumentation:

1. Prepare the data for the documents to publish by doing one or both of the following:

   - Create power-requiring cabinets or instruments by retrieving power elements from SmartPlant Instrumentation and running the tasks on the To Do List for the retrieved data. For details, see *Retrieve Data from the SmartPlant Instrumentation Interface*, page 455.

   - Create signals in SmartPlant Electrical for items that you want to associate with control systems in SmartPlant Instrumentation. For details, see *Creating Signals*, page 453.

2. Register a report template for each item type for which you want to publish data. For details, see *Register a Report*, page 413.

   💡 **Tips**

   - You can publish data for the following item types:

     - Cabinets owned by SmartPlant Instrumentation (the software identifies these using a database flag).

     - Instruments owned by SmartPlant Instrumentation (the software identifies these using a database flag).

     - Equipment (for both cabinets and instruments); the software does not publish other items types specified under Equipment. You can define your own document for publishing equipment data or you can use the shipped report, **Instruments and Cabinets.xls**, which is located in the folder where you install SmartPlant Electrical Reference Data.

- SignalRun. You can define your own document for publishing I/O signal data or you can use the shipped report, **IO Signal List.xls**, which is located in the folder where you install SmartPlant Electrical Reference Data. You can apply a filter to limit the scope of the signals to include, for example, only certain plant group types or certain I/O types.

Important

- When publishing data for item type Equipment, **do not** filter for cabinets or instruments. You can however, define a filter that limits the scope of the plant group type or the data subset.

- Select an existing filter to define the scope of the data to publish. If you require, you can register the same report template several times with different filters; however, each registered report must have a unique name. (For details of how to create or modify a filter, see Filter Manager Online Help.)

- You must select the **Enable document for publishing** check box to ensure that the software includes the data from the report in the generated .xml file when you publish from SmartPlant Electrical.

3. In the **Electrical Index**, expand the folder hierarchy **Documents** > **Reports**.

4. Select one or more of the reports for which you have enabled publishing, and click **SmartPlant** > **Publish**.

Tip

- You can also right-click the document, and then click **Publish**.

5. If you are working in File mode, on the **Publish to File** dialog box, specify the folder in which to save the generated .xml files by typing the path in the **Folder to save XML files** box and if you require, change the default data file and meta file names.

Related Topics
- *Creating Signals*, page 453
- *Prerequisites for Working with the SmartPlant Instrumentation Interface*, page 449
- *SmartPlant Instrumentation Interface: An Overview*, page 448

Publishing Data to an External Analyzing Tool

In SmartPlant Electrical you can generate single line diagrams that allow you to publish the data they contain to electrical analysis applications EDSA or ETAP. When you set the SLD generation options, you can specify the levels of the power distribution network for which the SLD displays the data, and this, in turn, affects the published output data. The software generates an .xml file in a format appropriate for the electrical analysis application, EDSA or ETAP, which you specify in the Options Manager. The data transfer process is 'one-way', that is, from SmartPlant Electrical to the electrical analysis application, according to the flow described in the following diagram:

After you publish the data from SmartPlant Electrical, both EDSA and ETAP have their own data exchange program which enables the import and translation of the SmartPlant Electrical .xml file for the appropriate electrical analysis application.

Related Topics
- *Publish Data to an External Analyzing Tool*, page 459

Publish Data to an External Analyzing Tool

1. In the Options Manager **General Settings** window, specify the external analyzing tool to which you want to publish data: ETAP or EDSA.

2. In the **Electrical Engineer**, select one or more nodes (starting items such as PDBs or buses) for which you want to generate your single line diagram.

3. Click **File** > **New** > **SLD** to open the **Single Line Diagram Options** dialog box.

4. On the **General** tab, type a document number and description.

5. Under **SLD type**, click **Electrical analysis SLD**.

6. On the **Circuits & Internals** tab, under **Options for electrical analysis SLDs**, click **Show only circuit internals**.

7. Enter additional data on the dialog box as required.

8. Click **Generate**.

9. With the SLD open, define a revision if required.

10. Click **File** > **Save**.

   💡 **Tip**

   • When you save the SLD, it appears automatically in the **Electrical Index** under **Documents** > **Electrical Analysis SLDs**.

11. Expand the hierarchy **Documents** > **Electrical Analysis SLDs** and select the document that you require.

   💡 **Tip**

   • If you made changes to the data that you want to include in the publish, first right-click the document and then click **Open**.

12. Right-click the document and then click **Publish to External Analyzing Tool**.

13. Specify the folder location for the .xml file containing the published data.

   📝 **Note**

   • After publishing the data from SmartPlant Electrical, you need to process the published file in EDSA or ETAP. For an outline of the process in the specific application, see:

   • *Process SmartPlant Electrical Data in EDSA*, page 463.

   • *Process SmartPlant Electrical Data in ETAP*, page 464.

Prerequisites and Mapping Requirements for Publishing Data to EDSA

✦ Important

- When using this interface, ensure that select lists in SmartPlant Electrical contain entries that match EDSA values. For example, transformer connections are mapped to Winding 1 type and the entries should be Delta and Star in both applications.

Prerequisites

The SmartPlant Electrical properties shown below require special treatment on the EDSA side.

Frequency

In SmartPlant Electrical, the Frequency property may have different values for various items on a particular branch. In EDSA, there is a master or project frequency, applied to all the EDSA buses on the branch. Whenever EDSA translator encounters a new frequency for a bus, it updates the master frequency with the frequency value of the last translated bus. Note that EDSA can support different frequencies for different branches.

Rated voltage

If a SmartPlant Electrical item does not have a rated voltage, EDSA uses the default system voltage, which is set to 13,800V.

In EDSA, each branch item has two voltages: a system voltage and a nameplate voltage. EDSA calculates the actual voltages for the non-voltage sources (branches and loads) while maintaining the system voltage and nameplate voltage as retrieved from SmartPlant Electrical.

SmartPlant Electrical select list values for rated voltages are converted and interpreted in EDSA as values with units of kilovolts.

Motor and generator synchronous speed

EDSA calculates this value based on the frequency and the number of poles of the equipment.

Description

This property is mapped to the EDSA **Description** property, however, in EDSA, the **Description** field is populated with data from more than one SmartPlant Electrical property. The various SmartPlant Electrical sources for descriptions are not concatenated; rather, the description value that appears in EDSA is the last non-null value translated from SmartPlant Electrical.

Cables

EDSA uses virtual cables to connect between retrieved SmartPlant Electrical branch items. These cables are identified by their Category: "<Connector>". They are not real cables, but rather connectors that the software needs to link between the two programs, and users should discard them.

EDSA cable length is calculated based on the two SmartPlant Electrical properties **Estimated cable length** and **Design cable length**. EDSA uses the design length if available; if not available (NULL), EDSA uses the estimated cable length. SmartPlant Electrical parallel cables are transformed into an EDSA cable with the correct number of parallel cables propagated into the EDSA **Cables/Phase** field.

Transformers

The following SmartPlant Electrical properties should be filled out and mapped to EDSA fields **X** and **R**, as percentages:

- Resistance Primary to Secondary 1
- Resistance Primary to Secondary 2
- Reactance Primary to Secondary 1
- Reactance Primary to Secondary 2
- Resistance Secondary to Secondary
- Reactance Secondary to Secondary

The SmartPlant Electrical property **NumberofSecondaries** defines whether EDSA creates a 2-winding or 3-winding transformer.

Data Mapping

The table below shows supported SmartPlant Electrical devices and their EDSA equivalents. For more complete and detailed information on mapping of tables and properties, please refer to the EDSA documentation.

SmartPlant Electrical Device	EDSA Equivalent Device
OffSitePower	Utility
Generator	Generator
Bus	Bus Bar
Motor	Induction Motor
Capacitor	Capacitor Bus
Heater	Static Load
Contactor	Breaker
HeatTrace	Static Load
HarmonicFilter	Static Load
Resistor	Static Load
BatteryCharger	not supported
BatteryBank	not supported
Instrument	Bus Bar
Cabinet	Static Load
OtherElectEquipment	Static Load
Cable	Feeder
Transformer	Transformer
Circuit	Feeder
DisconnectSwitch	Switch
CircuitBreaker	Breaker
ProtectionRelay	Breaker
OverloadRelay	Breaker
Fuse	Fuse
Starter	Circuit Breaker
OtherDisconnectEquip	Switch

📝 **Note**

- As indicated, some SmartPlant Electrical items are mapped to the same EDSA items, for example, various types of SmartPlant Electrical static loads are all mapped to the EDSA item type 'Static Load'.

Related Topics
- *Process SmartPlant Electrical Data in EDSA*, page 463
- *Publish Data to an External Analyzing Tool*, page 459
- *Publishing Data to an External Analyzing Tool*, page 457

Process SmartPlant Electrical Data in EDSA

🔷 **Important**

- For full details of this process, see the EDSA software documentation.

1. Run the EDSA Data Exchange program.

2. Choose one of the following commands:

   - **Import to EDSA**: this option creates a new EDSA database. You may specify the name of the EDSA database to create. If an EDSA database of that name already exists, it is discarded.

   - **Append to EDSA**: this option adds the entries from the SmartPlant Electrical project to an existing EDSA database. If a SmartPlant Electrical element already exists in the EDSA database, the new SmartPlant Electrical element overwrites the old EDSA element. If a new SmartPlant Electrical element does not yet exist in the EDSA database, the SmartPlant Electrical element is added to the EDSA database.

   💡 **Tips**

   - For each of these commands, the software automatically translates the file into a format suitable for importing the data into EDSA.

   - When using **Append**, there is no provision to determine that an element should be deleted from the EDSA database.

3. Open EDSA Technical 2000.

4. Click **File** > **Convert EDSA 2.95 File** to generate a drawing.

📝 **Notes**

- EDSA does not support the import of data for DC equipment such as battery banks, battery chargers, and DC buses.

- For 3-winding transformers, the software does not connect the secondaries in EDSA; you must make the connections manually.

- EDSA includes an error checking module that users can run to look for data errors that could cause problems in analysis programs, for example voltage mismatches between an item and the bus it is attached to.

- Normally, EDSA ignores SmartPlant Electrical properties with null values and uses whatever EDSA default values were already assigned to those properties. However, in EDSA, you can specify for each property a flag that enables a null value from SmartPlant Electrical to overwrite the default value in EDSA.

Related Topics
- *Prerequisites and Mapping Requirements for Publishing Data to EDSA*, page 459
- *Publish Data to an External Analyzing Tool*, page 459
- *Publishing Data to an External Analyzing Tool*, page 457

Process SmartPlant Electrical Data in ETAP

🔷 **Important**

- For full details of this process, see the ETAP software documentation.

1. Run the ETAP SmartPlant Data Exchange program.

2. In the **ETAP SmartPlant** dialog box, click **Browse** to navigate to the location of the SmartPlant Electrical .xml files for the project

3. Click **OK** to display the **Data Synchronization** dialog box, from which you can:

   - View and modify the data mapping.

   - Set defaults for the ETAP data.

   - Generate the data in a format suitable for ETAP.

4. In ETAP Power Station, open a new project.

5. Import the data using the .xml file generated by the ETAP SmartPlant Data Exchange program as the source.

Related Topics
- *Publish Data to an External Analyzing Tool*, page 459
- *Publishing Data to an External Analyzing Tool*, page 457

SmartPlant Electrical Project Management: An Overview

SmartPlant Electrical supports *Operating owner* and *EPC* activities with *As-Built* functionality. You define the main plant as the As-Built and you can then add any number of additional plant groups to create your projects. The Plant Administrator needs to activate the plant and *scope* it with the particular As-Built. When you work with *projects*, the database is partitioned into several schemas: a single schema for As-Built and separate schemas for projects. An operational plant exists and most of the activities are concerned with routine maintenance or plant modernization (revamps).

To facilitate plant modernization, the software enables you to create one or more projects using existing electrical data for the operating plant as a starting point for plant modernizations (revamps). Each project is defined for one plant only, and a plant can have several associated projects.

You can also use projects when working in SmartPlant environment.

After *merging* project data with As-Built, you cannot reverse the process. For this reason, at all stages of plant modernization, you should ensure that there is full coordination of engineering activities between As-Built and projects to avoid inadvertent loss of data. It is also recommended that you back-up your database before starting the projects.

📝 Notes

- You can work only in online mode — that is, As-Built and projects must be connected to the same database.

- When working in a plant that is registered with SmartPlant , the software automatically determines the project status in the database according to the SmartPlant project status.

- To be able to view and edit data in As-Built, make sure that in the Options Manager, on the **General Settings** page, the **Allow Full Access to As-Built** is set to **Yes**.

Related Topics
- *Project Management Common Tasks for a Project*, page 467

Project Management Common Tasks in As-Built

The following tasks are used frequently when working with As-Built in SmartPlant Electrical.

Select a Project

This procedure shows you how to select a project for the current As-Built plant. Note that you have to select a project every time you reopen the **Project Management** table. For more information, see *Select a Project*, page 478.

Modify Project Status

This procedure explains how to display the current project status, both with respect to SmartPlant and the plant and to update or modify those statuses. For more information, see *Modify Project Status*, page 474.

Add Items to the Project Management Table

This procedure explains how to open the **Project Management** table and add the items that you require. Note that you have to add items that you want to work with every time you reopen the **Project Management** table. For more information, see *Add Items to the Project Management Table*, page 484.

Scope Items

Once the Plant Administrator has scoped your project, you can start scoping the items that will be available for viewing in the project. For more information, *Scope Items*, page 481.

Use the Buffer to Scope Items

The buffer in the **Project Management** table allows you to make a preliminary selection of items that you want to scope. For more information, *Use the Buffer to Scope Items*, page 481.

Run a Test Merge

You can run a test merge before *merging* the items into As-Built. This operation allows you to check whether there are any problems with the items that you want to merge into As-Built. The software creates a log file that describes the problem that may occur during a test run. For more information, see *Run a Test Merge*, page 490.

Compare Documents

After scoping and modifying items in a project, you can compare documents of the same type in As-Built and in the project, for example, registered reports, SLDs, schematics. If changes that you made to the item are reflected in the document, the software compares the changes and indicates them on a comparison report. Note that the software generates As-Built and project reports automatically when you run this command. This comparison can serve as a precautionary measure before *merging* project documents into As-Built. For more information, see *Compare a Document in As-Built with a Document in a Project*, page 490.

Merge Items into As-Built

Merging is the final stage of a project cycle. After you have edited items in your project, you can merge those items into *As-Built*. The merge operation completely overwrites existing data in As-Built with data from the project. For this reason, once you have committed to merge data, the changes are irreversible. For more information, see *Merge Items into As-Built*, page 491.

Filter the Project Management Table Display

This option allows you to filter the display of the items in the **Project Management** table according to the status of items in the project. For more information, see *Filter the Project Management Table Display*, page 492.

Generate an Excel Report

This option allows you to generate a report in Excel showing the current selection in the **Project Management** table, arranged according to the main items. In the report, you can expand the main items to display their related items. For more information, see *Generate an Excel Report*, page 493.

Select a Display Option

This option allows you to specify the display mode in the data window. You can display the main items only, the main items expanded to show their related items, or a list showing all items. If a particular related item is associated with more than one main item, that related item appears once only in the list view. In list view only, you can sort the items as desired by clicking the column headers in the data window. For more information, see *Select a Display Option*, page 492.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Rules for Scoping Items*, page 478
- *Scoping Items: An Overview*, page 475
- *SmartPlant Electrical Project Management: An Overview*, page 465

Project Management Common Tasks for a Project

The following tasks are used frequently when working with a project in SmartPlant Electrical.

Add Items to the Project Management Table

This procedure explains how to open the **Project Management** table and add the items that you require. Note that you have to add items that you want to work with every time you reopen the **Project Management** table. For more information, see *Add Items to the Project Management Table*, page 484.

Claim Items

You *claim* items by opening the project to which they were scoped, and then you select individual items to claim. Note that you can claim main or related items separately. When claiming an item in a project, with the appropriate Options Manager setting, you can choose claim mode **Shared** or **Exclusive**. For more information, see *Claim Items*, page 484.

Set Claim Mode

This option allows you to set the required *claim* mode. The following claim modes are available:

- **Exclusive**— allows you to claim As-Built items in the current project only.

- **Shared**— allows users of different projects to claim the same As-Built items for their projects.

- **Release Claim**— allows you to cancel the claim of the selected items and then re-scope them in As-Built. Re-scoping the items will update the data in the project.

Note that the **Shared** claim mode is available only after making an appropriate setting in the Options Manager. Also, if your plant is registered with SmartPlant, the **Shared** mode is not available, regardless of the setting you define in the Options Manager. (For details, see General Settings Window in the *Options Manager User's Guide*). For more information, see *Set Claim Mode*, page 485.

Release Claim

Releasing the claim of an item enables you to cancel the claim of an item. This action makes it possible to re-scope the item in As-Built if you need to update the data in the project. After the item has been re-scoped, you can claim it again in the project and then edit the updated data. Note that when releasing the claim of an item, it status in the project remains **Claimed**. However, the claim mode changes to **Release Claim**. Items whose claim mode is **Release Claim** behave exactly in the same way as the scoped items as the software re-scopes these items. For more information, see *Release Claim*, page 485.

Mark Items as Completed

Just before you are ready return the items from your project to As-Built, you have to change the status of these items in the project to **Completed**. The items that have been marked as completed are no longer available for editing in the project and are ready for merging into As-Built. For more information, see *Mark Items as Completed*, page 486.

Clear Mark as Completed

This option allows to you to change the **Completed** status back to **Claimed**. These items will again become available for editing in the project. For more information, see *Clear Mark as Completed*, page 487.

Release Items from Merge

After merging items into the As-Built plant, users of other projects cannot claim these items. This option makes it possible to release merged items so that they can be claimed for other projects. After applying this option, the merged items in your current project become marked as scoped. For more information, see *Release from Merge*, page 487.

Filter the Project Management Table Display

This option allows you to filter the display of the items in the **Project Management** table according to the status of items in the project. For more information, see *Filter the Project Management Table Display*, page 492.

Generate an Excel Report

This option allows you to generate a report in Excel showing the current selection in the **Project Management** table, arranged according to the main items. In the report, you can expand the main items to display their related items. For more information, see *Generate an Excel Report*, page 493.

Select a Display Option

This option allows you to specify the display mode in the data window. You can display the main items only, the main items expanded to show their related items, or a list showing all items. If a particular related item is associated with more than one main item, that related item appears once only in the list view. In list view only, you can sort the items as desired by clicking the column headers in the data window. For more information, see *Select a Display Option*, page 492.

Related Topics

Scoping the Project

The Plant Administrator must specify those plant groups from which it will be possible to *scope* items. When the Plant Administrator specifies a plant group for scoping, the software includes in the scoping all higher-level plant groups in the same branch as the selected plant group, for example, if the plant groups are Plant-Area-Unit, and the Plant Administrator selects an area for scoping, the plant above that area is also included in the scope. For more information, see **Engineering Manager User's Guide, Set Project Scope**.

Related Topics
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Understanding Statuses

A **project** status shows the stage of the project life-cycle. Possible project statuses are:

- **Active**— this is the initial state of a project right after its creation.

- **Completed** — shows that all the work on the project items has been completed and that the items are ready to be *merged* back into As-Built.

- **Merged**— shows that all the project items have been merged back into As-Built.

- **Cancelled** — shows that the project has been cancelled and it can be deleted. Selecting this project status changes the status of the items in the project from **Claimed** to **Scoped**.

The status of **an item** in a project determines what you can do with the item, for example, editing the properties in the project. Possible item statuses in the project are:

- **Scoped** — shows that the item becomes available for viewing in the project and it can be *claimed*.

- **Claimed**— these are the scoped items that have been copied to the project and they are enabled for editing. There are two modes of claiming:

- **Exclusive**— these items can only be edited in the current project and cannot be claimed for another project.

- **Shared**— these items can be claimed for another project and edited in that project.

- **Completed**— shows that the work on the item has been completed and the item is ready to be merged back into As-Built.

- **Merged**— shows that the item has been merged back into As-Built.

The **Project Management** table displays the status of claimed items.

To view the status of a claimed item in a project while in As-Built, right-click an item in the list view pane of the **Electrical Index** and then click **Item Status in Project**. This command opens the **Item Status in Projects** dialog box, which shows the item status, claim mode, and project name for each project in which the selected item is claimed.

Related Topics
- *Modify Project Status*, page 474
- *Project Management Common Tasks in As-Built*, page 466
- *Rules for Changing Statuses*, page 472
- *SmartPlant Electrical Project Management: An Overview*, page 465

Rules for Changing Statuses

The following general rules apply to the relationship between the *project status* and *item status in project*:

1. To be able to scope items, the project status must be **Active** or **Merged**.

2. To be able to merge items, the item status in the project must be **Completed**.

3. You cannot re-scope an item (refresh the scope) if the item has already been claimed. You have to change the status of the item to **Scoped** and then reclaim it.

4. To be able to merge a project, all the items of that project must have status **Completed**.

5. It is possible to set the status of a project to **Completed** in As-Built if all the status of all claimed items is **Completed**.

6. In a project, it is possible to change the status of a project to **Completed** disregarding the status of the items in project. This operation automatically sets the status of all the items in the project to **Completed**, except for registered reports and all documents (SLDs, schematics, miscellaneous drawings and electrical analysis SLDs). To include these items when merging the project, you must mark them manually as completed in a separate session.

The following table shows the various commands available for changing the status of the items in the project and how they affect these statuses.

Command	Initial Status	Final Status	Comments
Scope	None, Scoped	Scoped	Performed in As-Built. If the initial status is **Scoped**, refreshes the data by re-scoping the item.
Set Claim Mode (Exclusive, Shared, or Release Claim)	Scoped	Claimed	Performed in a project. Invoked after clicking **Apply**.
Mark as Completed	Claimed	Completed	Performed in a project. Invoked after clicking **Apply**. Item stops being editable and is ready to be merged.
Clear Mark as Completed	Completed	Claimed	Performed in a project. Invoked after clicking **Apply** and receives same claim mode as when first claimed. Makes the item editable.

Command	Initial Status	Final Status	Comments
Mark as Reclaimed	Merged	Claimed	Performed in a project. Invoked after clicking **Apply** and receives same claim mode as when first claimed. Makes the item editable. This command is option is required when working with SmartPlant.
Release Claim	Claimed	Claimed	Performed in a project. Invoked after clicking **Apply**. The claim mode changes to **Release Claim**. Enables re-scoping of the item in As-Built.
Merge	Completed	Merged	Performed in As-Built. Overwrites the previous As-Built data.

Related Topics

Modify Project Status

1. Open the project for which you want to view the status.

2. On the **Project Management** toolbar, click 📇 to open the **Project Status** dialog box.

3. In the **SmartPlant Electrical project status** area, ascertain the current status of your project with respect to the Plant.

   💡 **Tip**

   - A check mark denotes the current status. Possible statuses include **Active**, **Completed**, **Merged**, and **Canceled**.

4. To change an active project to completed, click **Complete Project**. This button is available only when the **SmartPlant Electrical project status** is **Active**.

   💡 **Tip**

   - When you change the project status to **Completed**, the software automatically sets the status of all the items in the project to **Completed**, except for registered reports and all documents (SLDs, schematics, miscellaneous drawings and electrical analysis SLDs). To include these items when merging the project, you must mark them manually as completed in a separate session.

5. To change a completed project back to active, click **Return to Active**. This button is available only when the **SmartPlant Electrical project status** is **Completed**.

6. To merge a completed project into the Plant, click **Merge Project**. This button is available only when the **SmartPlant Electrical project status** is **Completed**.

7. To cancel the active project, click **Cancel Project**.

 SmartPlant Notes

- If you are registered with SmartPlant, click **Refresh Status** on the **Project Status** dialog box to update the **SmartPlant project status** display.

- In SmartPlant, the **Complete Project**, **Return to Active**, and **Merge Project** options are not available unless the **SmartPlant project status** is also active. The **Cancel Project** option is not available unless the **SmartPlant project status** is also **Canceled**.

Related Topics
- *Project Management Common Tasks in As-Built*, page 466
- *Rules for Changing Statuses*, page 472
- *SmartPlant Electrical Project Management: An Overview*, page 465

Scoping Items: An Overview

Once the Plant Administrator has *scoped* the project, you can start specifying which items will be available for viewing in the project. This selection is called scoping items.

You scope items by selecting the required items in As-Built. The selection can be one of the following:

- The entire plant hierarchy that was assigned in SmartPlant Engineering Manager, according to the scoping definitions that the Plant Administrator has made.

- Individual in the **Electrical Engineer**.

- Individual items in the **Electrical Index**.

- Several items in the Tabular Editor.

You can select loads, converting equipment, power sources, free cables, PDBs, and so forth. Note that the software automatically scopes all the items that are associated with the scoped items. These items are called related items. Related items are those items that have an electrical or functional relationship to a main item. Related items may include control stations, associated cables, circuits, and signals.

In addition to the items that are included in the scoped plant groups, you can also scope items that belong to unscoped plant groups. However, you will not be able to claim these items or edit them in the project.

When scoping, the software follows certain rules that govern which items become available for viewing in the project. For details, see *Rules for Scoping Items*, page 478.

Related Topics
- *Project Management Common Tasks in As-Built*, page 466
- *Rules for Scoping Items*, page 478
- *SmartPlant Electrical Project Management: An Overview*, page 465

Select a Project

1. In an As-Built plant, click **Window > New > Project Management**.

2. In the **Project Management** table, do one of the following:

   - Click on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click **Select Project**.

Notes

   - You have to select a project every time you reopen the **Project Management** table.

   - After closing the **Project Management** table, the software removes all the items from the **Project Management** table.

Related Topics
   - *Project Management Common Tasks in As-Built*, page 466
   - *SmartPlant Electrical Project Management: An Overview*, page 465

Rules for Scoping Items

The general rule for *scoping* is that when selecting items, the software automatically scopes all the related items and all the items that extend up to and including the item power source. For example, selecting a motor that is connected to a feeder circuit, the software will scope the motor control stations, the motor feeder cable, the feeder circuit, the bus, and the PDB.

The extent of the *related items* that are scoped along with the main selected item are indicated in the following table.

Main Item	Related Item	Notes
Loads, converting equipment, power sources (generators, battery banks, offsite power)	Cables of any category (from / to sides) Control stations and their cables Instruments and their cables I/O signals Associated Circuits, their internals and their PDB-Bus-Cell associations Metering equipment	Any cables connected to the circuit rather than directly to the scoped load will not be scoped with the circuit. Instead, those cables will be scoped with related equipment like control stations.
Power distribution boards	All buses, circuits and internals, up to the boundary of the PDB Metering equipment included as internals	No associated items external to the PDB items, such as cables or signals associated with the PDB or circuits will be scoped.
Drums		Does not include cables assigned to the drum that have not been scoped.

Main Item	Related Item	Notes
Cableways	All cableway segments Entire cable that is routed through one or more segments of the cableway	
Instruments	Cables of any category (from / to sides) I/O signals Associated Circuits, their internals and their PDB-Bus-Cell associations Metering equipment	
Local panels, junction boxes, cabinets	Cables of any category (from / to sides) I/O signals Associated Circuits and their internals Metering equipment	

🖉 Note

- You can scope the same item for more than one project.

Related Topics

Add Items to the Project Management Table in As-Built

1. In an As-Built plant, click **Window > New > Project Management**.

2. Select the desired project. For details, see *Select a Project*, page 477.

3. Drag the desired items one-by-one to the **Project Management** table. You can drag items from any of the following:

   - **Electrical Index**

   - **Electrical Engineer**

   - Tabular Editor

   - a schematic

   - a single line diagram

   📝 **Note**

   - You have to add items every time you reopen the **Project Management** table.

Related Topics
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Scope Items

1. In SmartPlant Electrical, click **File > Open > Plant Group** to open an As-Built plant.

2. On the **Open Plant Group** dialog box, click **Select Plant**.

3. On the **Open Plant Group** dialog box, select the appropriate plant hierarchy level and click **OK**.

4. Click **Window > New > Project Management**.

5. In the **Project Management** table in As-Built, click 🔄.

6. On the **Select Project** dialog box, select the project you require and click **OK**.

7. With the **Project Management** table in As-Built open, select the main items for scoping and then do one of the following:

   - Drag the selected item to the **Project Management** table. Note that you can only drag single items. For multiple items wherever possible, use the **Add** command.

   - Click 🖥 on the **Project Management** toolbar.

   💡 **Tip**

   - Note that you can use the buffer to make a preliminary selection of items that you may want to add to the **Project Management** tables. For details, see *Use the Buffer to Scope Items*, page 481.

8. Select the check box beside each main item that you want to scope.

9. Click 🔧 to scope the items.

📝 **Notes**

   - The software also scopes related items automatically.

   - Certain related items that are scoped, such as circuit internals, do not appear in the **Project Management** table.

Related Topics
   - *Project Management Common Tasks in As-Built*, page 466
   - *Rules for Scoping Items*, page 478
   - *Scoping Items: An Overview*, page 475
   - *SmartPlant Electrical Project Management: An Overview*, page 465

Use the Buffer to Scope Items

1. With the **Project Management** table in an As-Built plant open, do one of the following:

   - Click on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click **Show/Hide Buffer**.

2. Drag the desired items one-by-one to the **buffer** in the **Project Management** table. You can drag items from any of the following:

   - **Electrical Index**

   - **Electrical Engineer**

   - Tabular Editor

   - a schematic

   - a single line diagram

3. In the **Buffer**, click the check boxes next to the items that you want to scope.

4. Do one of the following:

   - Click on the on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click **Paste from Buffer**.

Related Topics

Claiming Items for Editing: An Overview

You *claim* items by opening the project to which they were scoped, and then you select individual items to claim. Note that you can claim main or related items separately. When claiming an item in a project, with the appropriate Options Manager setting, you can choose claim mode **Shared** or **Exclusive**.

📝 Note

- You do not need to claim new items that you create in the project.

Related Topics
- *Claim Items*, page 484
- *Project Management Common Tasks for a Project*, page 467

Add Items to the Project Management Table in a Project

1. Open the desired project.

2. Click **Window > New > Project Management**.

3. Do one of the following:

   - Drag a single item to the **Project Management** table from the **Electrical Index**, **Electrical Engineer**, or the Tabular Editor to the **Project Management** table.

   - Select a single item in a schematic or SLD and click 🖻.

   - Select multiple items the **Electrical Index**, or the Tabular Editor and click 🖻.

📝 Notes

- After closing the **Project Management** table, the software clears the all the items from the table.

- You have to add items every time you reopen the **Project Management** table.

Related Topics
- *Claiming Items for Editing: An Overview*, page 483
- *Project Management Common Tasks for a Project*, page 467
- *SmartPlant Electrical Project Management: An Overview*, page 465

Claim Items

1. In a project, click **Window > New > Project Management**.

2. Add the items that you require. For details, see *Add Items to the Project Management Table in a Project*, page 484.

3. Select the check box beside each item that you want to *claim*. For details of item statuses in the project that determine which items you can mark for claiming, see *Rules for Changing Statuses*, page 472.

4. For each item that you want to claim, select **Shared** or **Exclusive** as the claim mode. Note that if, in the Options Manager, **General Settings**, the **As-Built Claim Mode** option is set to **Exclusive**, the **Shared** claim mode is not available.

5. Click the **Apply** command to claim the items.

Notes

- The items are now available for editing in the project. When you have finished editing items, you can change their status to **Completed**. Items with status **Completed** are ready for merging into As-Built.

- Check the data under **Result Status** to make sure that there are no problems.

Related Topics
- *Claiming Items for Editing: An Overview*, page 483
- *Project Management Common Tasks for a Project*, page 467
- *SmartPlant Electrical Project Management: An Overview*, page 465

Set Claim Mode

1. In a project, click **Window > New > Project Management**.

2. Add the items that you require. For details, see *Add Items to the Project Management Table in a Project*, page 484.

3. Select the check box beside each required item. For details of item statuses in the project, see *Rules for Changing Statuses*, page 472.

4. Do one of the following:

- Click ⬇ on the **Project Management** toolbar and then select **Exclusive**, **Shared**, or **Release Claim**.

- Right-click in the **Project Management** table and then on the shortcut menu, point to **Set Claim Mode** and then click **Exclusive**, **Shared**, or **Release Claim**.

5. Click the **Apply** command to claim the items.

📝 **Note**

- The **Shared** claim mode is only available after selecting the appropriate setting in the Options Manager.

Related Topics
- *Claiming Items for Editing: An Overview*, page 483
- *Project Management Common Tasks for a Project*, page 467
- *SmartPlant Electrical Project Management: An Overview*, page 465

Release Claim

1. In the **Project Management** table (for a project), select the check box beside each main items for which you want to release the claim.

2. Do one of the following:

   - Click ✅ on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then on the shortcut menu, click **Release Claim**.

3. Click ☑ to release the claim for the selected items.

📝 **Notes**

- After setting the claim mode to **Release Claim**, the status of the items still remains **Claimed**. However, these items behave exactly in the same way as the scoped items as the software re-scopes them.

- To claim the items whose claim mode to **Release Claim**, right-click in the **Project Management** table and then on the shortcut menu, click **Mark as Reclaimed**.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Mark Items as Completed

1. Open the desired project.

2. Add the items that you require. For details, see *Add Items to the Project Management Table in a Project*, page 484.

3. Select the check box beside each item that you want to mark as completed. For details of item statuses in the project, see *Rules for Changing Statuses*, page 472.

4. Do one of the following:

   - Click 🖻 on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then on the shortcut menu, click **Mark as Completed**.

5. Click the **Apply** command to claim the items.

📝 **Note**

   - The items that have been marked as completed are no longer available for editing in the project and are ready for merging into As-Built.

Related Topics
   - *Claiming Items for Editing: An Overview*, page 483
   - *Project Management Common Tasks in As-Built*, page 466
   - *SmartPlant Electrical Project Management: An Overview*, page 465

Clear Mark as Completed

1. Open the desired project.

2. Add the items that you require. For details, see *Add Items to the Project Management Table in a Project*, page 484.

3. Select the check box beside each required item. For details of item statuses in the project, see *Rules for Changing Statuses*, page 472.

4. Do one of the following:

   - Click 🖋 on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then on the shortcut menu, click **Clear Mark as Completed**.

5. Click the **Apply** command to claim the items.

📝 **Note**

   - The items whose status is **Clear Mark as Completed** become available for editing in the project.

Release from Merge

1. In a project, click **Window > New > Project Management**.

2. Right-click in the **Project Management** table and then click **Show Merged Items**.

3. Select the check box beside each required item.

4. Do one of the following:

   - Click ![icon] on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click **Release from Merge**.

5. Do one of the following:

   - Click ![icon] on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click **Apply**.

✎ **Note**

   - After releasing from merge, the status of the items in the project becomes **Scoped**, that is, the items will be scoped after you click **Apply** again.

Related Topics
   - *Claiming Items for Editing: An Overview*, page 483
   - *Project Management Common Tasks for a Project*, page 467
   - *SmartPlant Electrical Project Management: An Overview*, page 465

Merging Items into As-Built: An Overview

Merging is the final stage of the project cycle. After you have edited items in your project, you can merge those items into *As-Built*. The merge operation completely overwrites existing data in As-Built with data from the project. For this reason, once you have committed to merge data, the changes are irreversible.

✎ Notes

- If you add a new item in the *project*, after merging the item, the software creates it in As-Built.

- If you modify data for an item in the project, after merging, the software overwrites the existing data for that item in As-Built.

- If you delete an item in the project, after merging the item, the software deletes it in As-Built.

- You can *claim* an item in a project and subsequently modify its data in As-Built. If you then want to transfer the item changes from As-Built to the project, you must update the item manually in the project. To assist you in doing this, it is recommended that you run a comparison report first. You can also run the **Release Claim** command in the project to automatically re-scope the item with the updated data from As-Built, but in this case any changes you have made to the item in the project will be lost.

- You can only edit data in As-Built if in the Options Manager, **General Settings**, the **Allow Editing in As-Built** property is set to **Yes**.

Related Topics

Compare a Document in a Project with a Document in As-Built

1. In an As-Built plant, click **Window > New > Project Management**.

2. Select the desired project. For details, see *Select a Project*, page 477.)

3. From the **Electrical Index**, add the desired documents to the **Project Management** table.   For details, see *Add Items to the Project Management Table in As-Built*, page 479.)

4. In the **Electrical Index**, select a registered report, a schematic or an SLD that you want to compare.

5. In the **Project Management** table, select the check box next to the document that you want to compare.

6. Right-click in the **Project Management** table and then click **Compare Documents**.

📝 Notes

- The revision numbering of a document in a project is done manually. The software does not raise the revision number automatically after a document was claimed and then changed in the project.

- In a multi-project environment, the software does not check the revision number of a claimed document in other projects even if the document was claimed for other projects.

- After merging the document into As-Built, the revision number of the document will be the one that was set in the project from which it was merged.

- In SLDs and schematics, the software displays clouds around the items that differ from the current data. The last revision number is shown in a triangle beside the cloud.

- The software uses index numbers to identify the changes in associations between electrical items.

- The software generates a tabular report where it shows all the changes and shows the changes in associations using index numbers as they appear in the drawing.

- For registered reports, the software opens two Excel files. One of the Excel files displays changed data with a blue shading. The second file is a summary of all changes and it is called **Registered Comparison Report**. This report displays the previous and current data for each tag that has undergone a change. Note that you can compare a registered report only if this is a simple tabular report.

- In SLDs, the software indicates the following changes:
- All added and deleted electrical items except for control stations.
- Added, deleted, and updated properties of electrical items.
- Changes in associations.
- Attachment to a different document template.
- In schematic drawings, the software indicates the following changes:
- Added, deleted, and updated properties of electrical items.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Run a Test Merge

1. In the **Project Management** table (for As-Built), select the desired project.
2. Do one of the following:

   - On the **Project Management** table toolbar, click .

   - Right-click in the **Project Management** table and then click **Show Completed Items** to see which items have been completed in the project.

3. Right-click in the **Project Management** table and then click **Test Merge**.

✎ **Note**

- The software generates a log file that shows any possible problems that might occur during the test merge. For details, see *Generate Error Log Files*, page 120.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Merge Items into As-Built

1. In an As-Built plant, click **Window > New > Project Management**.

2. Select the desired project. For details, see *Select a Project*, page 477.

3. Right-click in the **Project Management** table and then click **Show Completed Items** to see which items have been completed in the project.

4. Right-click in the **Project Management** table and then click **Merge**.

📝 Note

- The software merges all the completed items into As-Built, including the related items.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Filter the Project Management Table Display

1. In an As-Built plant or project, click **Window > New > Project Management**.

2. If your work environment is an As-Built plant, select the desired project. For details, see *Select a Project*, page 477.

3. Do one of the following:

   - Click the required icon on the **Project Management** toolbar.

   - Right-click in the **Project Management** table and then click one of the following commands:

   -

Icon	Command	Explanation
📊	**Show Completed Item**s	Filters the display to show only those items that have been completed in the project.
📊	**Show Merged Item**s	Filters the display to show only those items that have been merged back to As-Built.
📊	**Show Items Deleted from As-Buil**t	Filters the display to show only those items that have been deleted in As-Built.
📊	**Show Item Properties**	Displays the values of the item properties for an item similar to the **Properties** window view, but in read-only mode. The software can display the properties of only one item at a time; the item for which the row is highlighted.

Select a Display Option

1. In an As-Built plant or project, click **Window > New > Project Management**.

2. If your work environment is an As-Built plant, select the desired project. *Select a Project*, page 477.

3. Do one of the following:

    -   Click ▣ on the **Project Management** toolbar and then select the required option.

    -   Right-click in the **Project Management** table, click **Display**, and then click a command: **Main Items Only**, **Main Items with Related Items**, **All Items as List**.

Related Topics
-   *Project Management Common Tasks for a Project*, page 467
-   *Project Management Common Tasks in As-Built*, page 466
-   *SmartPlant Electrical Project Management: An Overview*, page 465

Generate an Excel Report

1. In an As-Built plant or project, click **Window > New > Project Management**.

2. If your work environment is an As-Built plant, select the desired project. *Select a Project*, page 477.

3. Do one of the following:

    -   Click ▤ on the **Project Management** toolbar.

    -   Right-click in the **Project Management** table and then click **Excel Report**.

Related Topics
-   *Project Management Common Tasks for a Project*, page 467
-   *Project Management Common Tasks in As-Built*, page 466
-   *SmartPlant Electrical Project Management: An Overview*, page 465

Backup a Project

1. In an As-Built plant or project, click **Window > New > Project Management**.

2. If your work environment is an As-Built plant, select the desired project. For details, see *Select a Project*, page 477.

3. In the **Project Management** table, do one of the following:

   - Click ⊟ on the **Project Management** toolbar and select **Backup**.

   - Right-click in the **Project Management** table and then click **Backup / Restore > Backup**.

4. On the **Create New MDB File for Backup** dialog box, type the name of the file select a desired location.

✎ Notes

- The software saves the entire data in the current project to a Microsoft Access database (.mdb) file.

- Project backup in SmartPlant Electrical is compliant with SmartPlant backup and restore instructions.

Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Restore a Project

Important

- When restoring a backed-up project, the software completely overwrites the data of the current project with the data in the Microsoft Access database (.mdb) file that you select. Since this process is irreversible, please exercise extreme caution before restoring a project.

1. In an As-Built plant or project, click **Window > New > Project Management**.

2. If your work environment is an As-Built plant, select the desired project. *Select a Project*, page 477.

3. Do one of the following:

   - Click on the **Project Management** toolbar and select **Restore**.

   - Right-click in the **Project Management** table and then click **Backup / Restore > Restore**.

4. On the **Select MDB File for Restore** dialog box, a desired Microsoft Access database (.mdb) file and click **Open**.

Note

- Project restore in SmartPlant Electrical is compliant with SmartPlant backup and restore instructions.
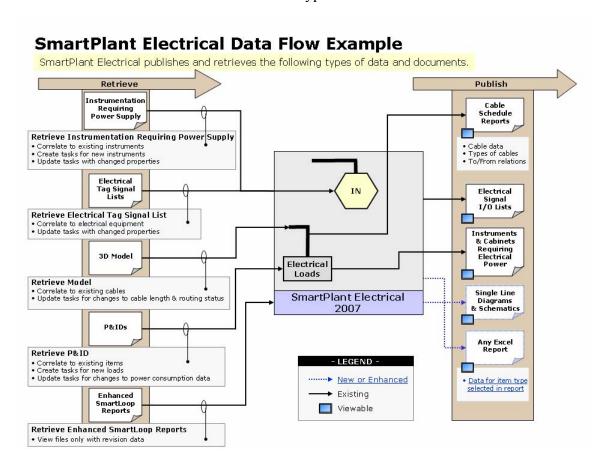
Related Topics
- *Merging Items into As-Built: An Overview*, page 488
- *Project Management Common Tasks in As-Built*, page 466
- *SmartPlant Electrical Project Management: An Overview*, page 465

Working with SmartPlant Integration: An Overview

SmartPlant integration standardizes and improves the communication among the various authoring tools you use in the course of designing, constructing, and operating a plant. SmartPlant Foundation acts as a repository for data and a medium through which information is shared among other tools, such as SmartPlant Electrical, SmartPlant Instrumentation, and SmartPlant P&ID, SmartPlant 3D, and Zyqad (does not interface with SmartPlant Electrical).

Most of the commands that provide access to SmartPlant integration functionality exist in the common user interface available on the **SmartPlant** menu in SmartPlant Electrical.

The following graphic displays what SmartPlant Electrical publishes and retrieves and shows the flow of data and the different types of data.

SmartPlant Electrical interacts with SmartPlant Foundation by correlating items between the plant database and the SmartPlant Foundation database, retrieving such documents as Electrical Power Elements, Electrical Signal List, Cable Schedule reports, and P&IDs, and the like from SmartPlant Foundation. Also, SmartPlant Electrical creates a set of tasks in the **To Do List** that you can run to update the plant database. In SmartPlant Electrical, you can also use the **SmartPlant** menu to publish documents and retrieve data, access the SmartPlant Foundation Web Client in order to browse in SmartPlant Foundation, and subscribe to change notifications and compare documents.

You can only use the **SmartPlant** menu commands after your plant is registered. For more information, see *SmartPlant Engineering Manager Help* and **Related Topics**.

Registering Tools

Before you can publish and retrieve documents from any of the other authoring tools, such as SmartPlant P&ID or SmartPlant Instrumentation, you must register each plant in SmartPlant Electrical with a SmartPlant Foundation database. The connection allows SmartPlant Electrical to use the commands on the **SmartPlant** menu. A SmartPlant Engineering Manager administrator typically registers a plant.

The software maps a plant and all its projects to a single SmartPlant Foundation URL, which points to one, and only one, SmartPlant Foundation plant database and its projects. When you use the **Register** command in any of the authoring tools, you are registering an authoring tool plant with a SmartPlant Foundation URL and plant that you specify.

The system administrator must register each plant in the authoring tool once; this action takes place in SmartPlant Engineering Manager. After the plant is registered, you can publish and retrieve documents.

Related Topics
- *Access the SmartPlant Foundation Web Client*, page 498
- *Document Publish and Retrieve Matrix*, page 502
- *Integrating SmartPlant Electrical*, page 496
- *Publishing: An Overview*, page 517
- *Retrieving: An Overview*, page 522
- *Schema Mapping: An Overview*, page 503

Integrating SmartPlant Electrical

This topic describes rules and settings that allow SmartPlant Electrical data to be shared correctly with SmartPlant 3D, SmartPlant Instrumentation and SmartPlant P&ID, and the other tools that are part of an integrated environment. Other tools that are not listed here have no known SmartPlant Electrical / SmartPlant integration issues.

General Requirements for a SmartPlant Electrical Plant with Projects

To use a plant that has projects in an integrated environment, you need to make the following changes in SmartPlant Electrical Options Manager, in the **General Settings** window:

- For the **Allow Full Access to As-Built** setting, select **No**. To use such a plant in an integrated environment, you must ensure that it is not possible to change As-Built data in As-Built itself. After you select **No** for this setting, the software can only update As-Built when you merge project data with As-Built.

- For the **As-Built Claim Mode** setting, select **Exclusive**. SmartPlant integration does not support having the same item claimed for different projects.

♦ Important

- When SmartPlant P&ID and SmartPlant Electrical are both enabled for a plant, they are both enabled for all projects of that plant. If a project requires only one of these applications, create separate plants for each application, and then enable SmartPlant P&ID for one plant and SmartPlant Electrical for the other.

Integrating SmartPlant Electrical with SmartPlant Instrumentation

In order to have a successful data exchange between SmartPlant Electrical and SmartPlant Instrumentation, make sure that:

- Both SmartPlant Electrical and SmartPlant Instrumentation have the same plant hierarchy structure

- In SmartPlant Instrumentation, instruments and I/O signals are defined on the lowest plant hierarchy level

- In SmartPlant Instrumentation, cabinets are defined on the highest plant hierarchy level

- In SmartPlant Electrical, you defined a registered report for both the I/O lists and for Instruments requiring power supply.

You must also ensure that the following SmartPlant Electrical select lists and SmartPlant Instrumentation supporting tables contain identical values:

- System I/0 type
- Rated Voltage
- Power Supply AC/DC Flag
- Number of Phases
- Operating Mode
- Process Function
- Frequency

Furthermore, in SmartPlant Electrical Options Manager, in the **General Settings** window, you need to make the following changes:

- For the **SmartPlant Instrumentation Default Domain** setting, specify the name of a SmartPlant Instrumentation domain which you want to set as a source domain for retrieving detailed wiring data to be included in schematics.
- If you want to retrieve enhanced reports to be included in schematics, select **Yes** for the setting **Include Enhanced Reports in Schematics**.

In the two tools, the units of measure of certain properties might be defined using a different precision accuracy. For example, 1.2 kW (one-digit accuracy of precision) in SmartPlant Electrical corresponds to 1.23 kW (two-digit accuracy of precision) in SmartPlant Instrumentation. This may result in inconsistencies and could be interpreted as an update when retrieving data.
These properties are:

- Rated Active Power
- Rated Reactive Power
- Rated Apparent Power
- Full Load Current
- Operating Mode Coincidence Factors (X, Y, Z, ZZ)
- Starting Current

Integrating SmartPlant Electrical with SmartPlant 3D

In order to have a successful data exchange between SmartPlant Electrical and SmartPlant 3D, make sure that SmartPlant Electrical reference cables used in the project match the cables SmartPlant 3D has in its Catalog. SmartPlant 3D identifies the type of the retrieved cable by looking at the reference cable name in the Catalog; if this cable in the Catalog does not exist, the retrieved cable is identified as an unknown cable for cable routing purposes.

Notes

- SmartPlant Electrical ships a sample report named Reference Data Cables Interface to SmartPlant 3D, that you can use as a source to be transferred to SmartPlant 3D, so that SmartPlant 3D users can create their Catalog using the report data.

- If cables that originated in SmartPlant Electrical were subsequently deleted in SmartPlant 3D and then published back from SmartPlant 3D, SmartPlant Electrical does not generate **Delete** tasks for those cables.

Make sure that the following select lists in SmartPlant Electrical must match the content of the related select lists in SmartPlant 3D:

- Routing status
- Segregation level

SmartPlant Electrical also publishes to SmartPlant 3D the relations to the cable side 1 and side 2. If SmartPlant 3D does not contain the relations, SmartPlant 3D cannot perform cable routing.
This means that:

- In SmartPlant Electrical, the cables must have their side 1 and side 2 associated before publishing.

- In SmartPlant 3D, users must make sure that each object referenced in the cable sides has a corresponding object in the SmartPlant 3D data model.

Integrating SmartPlant Electrical with SmartPlant P&ID

SmartPlant Electrical can retrieve items of electrical equipment that were published from SmartPlant P&ID. Electrical equipment in SmartPlant P&ID constitutes electrical motors and heaters only.

SmartPlant Electrical retrieves any items of process equipment that are associated with the electrical equipment: for example, if a motor is associated with a pump in SmartPlant P&ID, the pump is retrieved along with the motor in SmartPlant Electrical. Likewise, if the software retrieves a heater that is associated with a vessel, then that vessel is also retrieved.   The software also supports retrieval of multiple electrical equipment associations to a single item of process equipment, for example, where several heaters are used to heat up one vessel. Note that SmartPlant Electrical does not retrieve items of process equipment that are not associated with electrical items.

Note

- When retrieving an item of mechanical equipment associated with a motor from SmartPlant P&ID, the numerical value and units of measure of the **Power Driver Rated** property as set in SmartPlant P&ID must exactly match an existing value of the **Motor Rated Power** property in SmartPlant Electrical for the task to run successfully. In addition, if you run the SmartPlant Electrical command to copy the value of the process equipment **Absorbed Power** property to the motor **Brake Power** property, the software only validates values of the brake power that do not exceed the motor rated power, as determined by the standard SmartPlant Electrical rule.

Related Topics
- *Publishing: An Overview*, page 517
- *Retrieving: An Overview*, page 522
- *Working with SmartPlant Integration: An Overview*, page 495

Access the SmartPlant Foundation Web Client

- Click **SmartPlant > Browser**.

💡 Tip

- This command is available only if the active plant has been registered. For more information, see *SmartPlant Engineering Manager Help* and **Related Topics**.

✎ Note

- From the SmartPlant Foundation Web Client, you can perform a number of tasks, such as publishing or retrieving documents, comparing documents, subscribing to document changes, and so forth. Many of these tasks can be performed from the authoring tools, such as SmartPlant Instrumentation or SmartPlant P&ID, but the Web Client provides unique access to other features such as the Web Client To Do List and search capabilities.

Related Topics
- *Integrating SmartPlant Electrical*, page 496
- *Schema Mapping: An Overview*, page 503
- *Working with SmartPlant Integration: An Overview*, page 495

Document Publish and Retrieve Matrix

The following table lists authoring tools that participate in integration and the documents that those authoring tools publish and retrieve.

Authoring Tool	Publishable Documents	Retrieved By					
Zyqad	**SmartPlant P&ID**	**SmartPlant Electrical**					
Zyqad	Equipment Datasheets		X				
Equipment Lists (published as Equipment Datasheets)		X					
Process Flow Diagrams		X					
Stream Datasheets (published as Equipment Datasheets)		X					
SmartPlant P&ID	P&IDs	X		X		X	X
SmartPlant Electrical	Cable Schedule Reports						X
SPEL Electrical Power Elements Reports					X		
SPEL Electrical Signal List Reports					X		
SmartPlant Instrumentation	Browsers (Viewable Files)						
Dimensional Datasheets (DDP)						X	
Instrument Index Documents		X					
Instrument Loop Diagrams							
Instrument Process Datasheets							
Instrument Specification Sheets							
SPI Electrical Power Elements Reports			X				
SPI Electrical Signal List Reports			X				

Authoring Tool	Publishable Documents	Retrieved By					
SmartPlant 3D	3D Model Data (SmartPlant Review File Type)						
3D Model Data filtered for cable schedule data (SmartPlant Review File Type)		X					
Drawings (Viewable File with Links to Data)							
Reports (Viewable File with Links to Data)							
PDS	Drawings						
Model							
Reports							
SmartPlant Foundation	Equipment Lists	X					
Instrument Process Datasheets				X			
Instrument Master Lists		X					
Plant Breakdown Structure (PBS)	X			X	X	X	
Project Breakdown Structure						X	
Project Definition Documents			X	X			
Project Lists	X					X	

![icon] **Notes**

- SmartPlant Engineering Manager retrieves plant and project information for SmartPlant P&ID and SmartPlant Electrical.

- SmartPlant Review is integrated with SmartPlant Foundation for interactively reviewing 3D models.

Related Topics
- *Integrating SmartPlant Electrical*, page 496
- *Publishing: An Overview*, page 517
- *Retrieving: An Overview*, page 522
- *Working with SmartPlant Integration: An Overview*, page 495

Schema Mapping: An Overview

This section describes the assumptions, rules, and limitations imposed by the SmartPlant Electrical adapter on schema mapping.

This section describes the structure of the tool schema and how it is used by the authoring tool SmartPlant integration adapter in support of the publish and retrieve operations. This topic does not provide a list of the classes, properties, or enumerated lists (select lists) that are mapped by the delivered SmartPlant Electrical tool schema. You can obtain this information by viewing the tool schema directly. This set of topics does not describe the user interface of the Schema Editor.

Before modifying the mapping for SmartPlant Electrical, you must understand:

- The SmartPlant Electrical software
- The SmartPlant Electrical data model
- The delivered mapping for SmartPlant Electrical
- How the delivered mapping for SmartPlant Electrical works
- SmartPlant schema modeling
- The Schema Editor
- The To Do List feature of SmartPlant Electrical

In the topics that describe schema mapping, the following terms are frequently referenced:

- Tool schema — Defines the SmartPlant Electrical objects to be published or retrieved and how they map to objects in the SmartPlant schema. The tool schema implements mapping between the objects in the SmartPlant Electrical data model and the objects in the SmartPlant schema. Users who customize either the SmartPlant Electrical data model or the SmartPlant schema may also need to customize the mapping contained in the tool schema.
- SmartPlant schema — Defines the SmartPlant integration data model.
- Adapter — The SmartPlant Electrical code module that supports the SmartPlant integration commands.

When you publish SmartPlant Electrical data, the SmartPlant integration adapter extracts data from the SmartPlant Electrical database and converts it into a set of objects and relationships that are compatible with the SmartPlant schema. These objects and relationships are defined in an .xml file, which is sent to SmartPlant Foundation for further processing. For SmartPlant Electrical, the name of this file is SPELMap.xml, and it is located in the SmartPlant Electrical home folder.

General Information About Schema Mapping

This topic provides general information about the mapping in SmartPlant Electrical.

SmartPlant Electrical Adapter

Each authoring tool has an adapter that processes information during the publish and retrieve operations. One of the adapter's functions is to map information between the SmartPlant Foundation database and the particular authoring tool. The SmartPlant Electrical adapter is a component that belongs to SmartPlant Electrical and requires installation of SmartPlant Electrical Version 3.5 on one of the following database platforms: Oracle 8i, 9i, or 10g, or SQL Server 2000. Any communication between SmartPlant Foundation and SmartPlant Electrical is performed via this adapter, which sends information to SmartPlant Foundation through the Common UI where the user is shown the appropriate user interface and data available.

The adapter does not handle tool schema files directly, but uses them indirectly. The tool schema files are used to define the retrieve map file, which is then used by the adapter. Direct use of the schema files would mean that the adapter accessed them directly to do its work. Both retrieve and publish operations only work through map files.

Mapping Configuration

The SmartPlant Electrical adapter uses the tool schema map file and the Schema Editor, delivered with the SmartPlant Schema Component, to perform mapping between the SmartPlant Electrical object structure and the structure of objects in the SmartPlant schema. The map file is installed with SmartPlant Electrical Setup. This file includes all objects that are mapped for SmartPlant (not all objects that exist in SmartPlant Electrical, not all properties, only those that can be of any relevance for other applications that use SmartPlant). In accordance with the SmartPlant Electrical data model, the Schema Editor knows what SmartPlant Electrical object to put a property on.

Tool Schema Location

The name of the SmartPlant Electrical tool schema file is SPELMap.xml. SmartPlant Electrical uses this file for all types of published and retrieved documents. This file must exist in the installation home folder of SmartPlant Electrical. The SPELMap.xml should be treated as an installation file. Although, using the Schema Editor, it is possible to change the content of the map file, the file name must remain SPELMap.xml.

Related Topics

Mapping Rules and Limitations

Schema mapping in SmartPlant Electrical has some important rules and limitations.

Rules for Mapping Documents and SmartPlant Foundation Revisions

In SmartPlant Electrical, a document object has various properties but only the document type property is mapped to a document category enumeration list in the SmartPlant schema. A list of SmartPlant Electrical documents that can be published and retrieved is defined in the SmartPlant schema and can be viewed in the Schema Editor.

Also, SmartPlant Electrical adapter supports mapping of SmartPlant Foundation revisions. Whenever a SmartPlant Electrical user creates a new revision, the SmartPlant Client calls a common dialog box in which the user defines the revision schema and the revision number, this number is saved in SmartPlant Foundation and SmartPlant Foundation returns this number to SmartPlant Electrical via the adapter.

Rules for Property Naming

The names of the properties in the SmartPlant Electrical tool schema must exactly match the names of the properties in the SmartPlant Electrical data model, including the case.

Limitations of the SmartPlant Schema

The retrieve map is updated directly from the contents defined in the SmartPlant schema; therefore, the following restriction on the SmartPlant schema exists: in rare cases, if an entry is removed from the SmartPlant schema, you may need to edit the retrieve map file to remove the deleted entry. You should use the Schema Editor delivered with the SmartPlant Schema Component for this purpose.

Map File Location and Definition Restrictions

You cannot change the location of the map file SPELMap.xml, installed in the SmartPlant Electrical home folder. During publish and retrieve operations, the adapter looks for the map file only in the SmartPlant Electrical home folder. Although, using the Schema Editor, it is possible to change the content of the map file, the file name must remain SPELMap.xml.

Equipment Inheritance Mapping Restrictions

Currently, SmartPlant Electrical equipment inheritance is not implemented in the tool schema. This is a subject of future development. You have to map properties of equipment manually. Since there is no equipment inheritance, mapping is only performed on the level of a concrete (tangible) object class. To define relationships between various object classes, interfaces in the tool schema are used. Interfaces in the tool schema do not contain properties.

Objects and Relationships

This topic describes objects and object relationships that are relevant to the SmartPlant Electrical tool schema. Relationship descriptions reference names shown in a diagram that represents the data model of the SmartPlant Electrical tool schema. For details, see Tool Schema Data Model Diagram.

MapClass

The MapClass object corresponds to the IMapClassDef interface in the Tool Schema Data Model diagram. In general, there is a MapClass object in the tool schema for each SmartPlant Electrical class (ItemType) that needs to be published or retrieved. MapClass is referred to as SPMapClassDef in the Schema Editor user interface. The properties and relationships for a MapClass object can be viewed in the **Edit Map Class Definition** dialog box.

General

Property	Description
UID	The property value is required, must be unique, with all characters in upper case. UID of a map class must be SPEL_<CLASSNAME>, where CLASSNAME is the name of a corresponding item type in the SmartPlant Electrical data model. For example, the UID of the Motor class must be SPEL_MOTOR.
Name	The property value is required. This property designates the internal name of the SmartPlant Electrical item type, exactly as defined in the Data Dictionary.

Tool Schema Relationships

Relationship	Description
Map Properties	The map properties associated with the MapClass object must be defined and related to the MapClass object. The relationship name in the diagram is MapClassMapProperties

Mapping Relationships

Relationship	Description
Publish to SmartPlant Class	This relationship must be set to a class in the SmartPlant schema if this class is to be published. When this relationship is set, an instance of the target class is published for each instance of the source class in the document being published. The relationship name in the diagram is MapClassToClass.
Retrieve from SmartPlant interface	The list of interfaces from which properties are to be retrieved must be set if this class is to be used for retrieval. The relationship name in the diagram is ClassToMapClass.

MapProperty

The MapProperty object corresponds to the IMapPropertyDef interface in the Tool Schema Data Model diagram. In general, there is a MapProperty object in the tool schema for each SmartPlant Electrical property that needs to be published or retrieved. The properties and relationships for a MapProperty object can be viewed in the **Edit Map Property Definition** dialog box.

General

Property	Description
UID	The property value is required, must be unique, with all characters in upper case. UID of a map property must be SPEL_<CLASSNAME>_<PROPERTYNAME>. For example, UID of Motor.ItemTag property must be SPEL_MOTOR_ITEMTAG.
Name	The property value is required. This property designates the internal name of the SmartPlant Electrical attribute, as defined in the Data Dictionary.

Property	Description
Process Criteria	Process criteria are used when retrieving properties of SmartPlant Electrical objects in the following case: 1. In SmartPlant Electrical, there is an object mapped to an object in the SmartPlant schema, for example **SPEL_A** to **SmartPlant_A**. 2. Not all of the SmartPlant Electrical object properties are mapped to the properties of the mapped object in the SmartPlant schema. For example, **SPEL_A_Property_A** is mapped to **SmartPlant_A_Property_A** but **SPEL_A_Property_B** is not mapped in the **SmartPlant_A** object because **SmartPlant_A** does not contain the corresponding property **SmartPlant_A_Property_B**. 3. The corresponding property is available in another SmartPlant schema object, for example, **SmartPlant_AA**. To map the **SPEL_A_Property_B**, a relationship must exist between the **SmartPlant_A** and **SmartPlant_AA** objects. A relationship is designated by a text string that corresponds to the process criteria setting of the SPEL_A_Property_B property in SmartPlant Electrical. If there are intermediate objects between the mapped SmartPlant schema object and the object in the SmartPlant schema that has the required property, the process criteria of the property in SmartPlant Electrical must contain all relationship definitions of SmartPlant schema objects, separated by a comma delimiter.

Tool Schema Relationships

Relationship	Description
Exposed by Map Classes	Every MapProperty object must be related to a MapClass object. The relationship name in the diagram is MapClassMapProperties.
Scoped by Enumeration	If the data type of the property is enumerated, the MapProperty object must be related to a MapEnumList object. If the data type of the property is UoM, the MapProperty must be related to a MapUoMList object. Otherwise, this relationship is not needed. The relationship name in the diagram is MapPropertyMapEnumList.

Mapping Relationships

Relationship	Description
Map Property to Property	This relationship defines the property in the SmartPlant schema to which this tool property is to be published. If this relationship is not defined, the property will not be published. The relationship name in the diagram is MapPropertyToProperty.
Property to Map Property	This relationship defines the property in the SmartPlant schema from which this tool property is to be retrieved. If this relationship is not defined, the property will not be retrieved. The relationship name in the diagram is PropertyToMapProperty.

MapEnumList

The MapEnumList object corresponds to the IMapEnumListDef interface in the Tool Schema Data Model diagram. In general, there is a MapEnumList object in the tool schema for each SmartPlant Electrical enumerated list associated with a property that needs to be published or retrieved. The properties and relationships for a MapEnumList object can be viewed in the **Edit Map Enumerated List Definition** dialog box.

General

Property	Description
UID	The property value is required, must be unique, characters are not case-sensitive. UID of an enumerated list must be SPEL_<EnumListName>. For example, UID of the YesNo enum list must be SPEL_YesNo.
Name	The value of this property is the name of the SmartPlant Electrical enum list.

Tool Schema Relationships

Relationship	Description
Scoped Map Properties	A MapEnumList object must be related to the MapProperty objects that it scopes. The relationship name in the diagram is MapPropertyMapEnumList.
Contains	A MapEnumList object can contain a variable number of MapEnum objects. The relationship name in the diagram is MapEnumListMapEnum.

Mapping Relationships

Relationship	Description
Maps To	To support the publish operation, a MapEnumList object should be related to the corresponding enum list in the SmartPlant schema by means of the Maps To relationship. The relationship name in the diagram is MapEnumListToEnumList.
Maps From	To support the retrieve operation, a MapEnumList object should be related to the corresponding enum list in the SmartPlant schema by means of the Mapped From relationship. The relationship name in the diagram is EnumListToMapEnumList.

MapEnum

The MapEnum object corresponds to the IMapEnumDef interface in the Tool Schema Data Model diagram. In general, there is a MapEnum object in the tool schema for each SmartPlant Electrical enumerated list entry that needs to be mapped. The properties and relationships for a MapEnum object can be viewed in the **Edit Map Enumerated List Definition** dialog box (the same dialog box is used for MapEnumList objects.)

General

Property	Description
UID	The property value must be unique. UID of an enumerated list entry must be SPEL_<EnumListName>_<CodelistIndex>, where CodelistIndex is the value of the index of the corresponding select list entry in the SmartPlant Electrical data model. Values of codelist index are stored in the Data Dictionary table CODELISTS. For example, UID of the Yes entry of the YesNo enum list must be SPEL_YesNo_1.
Name	The value of this property is a text setting of the SmartPlant Electrical enum list entry.

Tool Schema Relationships

Relationship	Description
Contained In	Every MapEnum object must be related to a MapEnumList object. The relationship name in the diagram is MapEnumListMapEnum.

Mapping Relationships
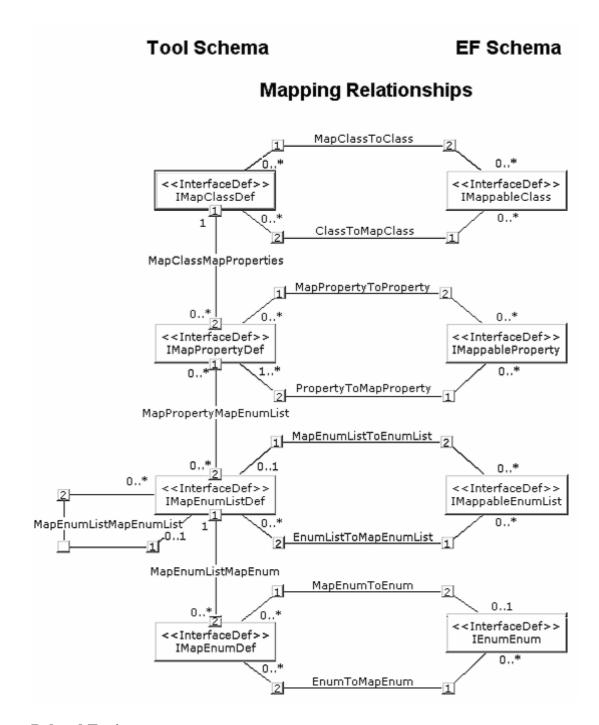
Relationship	Description
Maps To	This relationship defines the SmartPlant schema enum value to which this SmartPlant Electrical enum value is to be published. If this relationship is not defined, the enum value will not be published. The relationship name in the diagram is MapEnumToEnum.
Maps From	This relationship defines the SmartPlant Electrical enum value to which the SmartPlant schema enum value is to be retrieved. If this relationship is not defined, the enum value will not be retrieved. The relationship name in the diagram is EnumToMapEnum.

Related Topics
- *General Information About Schema Mapping*, page 505
- *Mapping Rules and Limitations*, page 506
- *Schema Mapping: An Overview*, page 503

Tool Schema Data Model Diagram

The diagram below shows a representation of the objects and relationships that are relevant to the SmartPlant Electrical tool schema. The rectangles on the left side of the diagram represent objects within the tool schema. The rectangles on the right represent objects within the SmartPlant schema. The relationships that connect objects on the left with objects on the right are the mapping relationships. The map file contains both the tool schema objects and the mapping relationships.

Tool Schema

EF Schema

Mapping Relationships

Related Topics
- *General Information About Schema Mapping*, page 505
- *Objects and Relationships*, page 507

Publish Operation Description

Documents that SmartPlant Electrical can publish are reports saved in Excel format. A SmartPlant Electrical report contains an item list responding to certain criteria. Items in the list can have relationships with other types of items that are not listed in the report. A published document contains both the report items and the related items. The underlying logic of defining relationships between the items in the SmartPlant Electrical data model is partly hard-coded in SmartPlant Adapter.

When a SmartPlant Electrical document is published, SmartPlant Adapter extracts data from the SmartPlant Electrical database and converts it into a set of objects and relationships that are compatible with SmartPlant schema. These objects and relationships are written out to an .xml file and are sent to the SmartPlant server for further processing. SmartPlant Adapter depends on the data in the tool schema to support the publish operation. For each published class in the tool schema, there is a code module in SmartPlant Adapter known as a Map Object. The Map Object contains code for the following tasks:

- Create a Data Object — For each item to be published, the Map Object creates a data object in the document container. The class of this object is the SmartPlant Class related to the MapClass via the MapClassToClass relationship.

- Set Property Values — The property values on the data object are set according to the values on the SmartPlant Electrical item. The list of properties to be published is defined by the list of MapProperties related to the current MapClass.

- Publish Relationships — The Map Object defines the types of relationships that are to be published for each data object it publishes.

Related Topics
- *General Information About Schema Mapping*, page 505
- *Mapping Rules and Limitations*, page 506

Retrieve Operation Description

When an external document is retrieved, SmartPlant Adapter compares the objects in that document to the data in the SmartPlant Electrical database and generates the following tasks in the To Do List:

- Create Task — Generated when a new item needs to be created
- Update Task — Generated when an existing item needs to be modified
- Delete Task — Generated when an existing item needs to be deleted

SmartPlant Adapter depends on the data in the tool schema to support the retrieve operation. When a document is retrieved, SmartPlant Adapter receives two containers of data:

- Document Container — Includes all of the published objects and relationships for that document
- Tombstone Container — Includes information about deleted objects and relationships

SmartPlant Adapter Actions in the Document Container

SmartPlant Adapter iterates over all of the objects in the Document Container and performs the following actions:

- Use Primary Interface to Determine Class — The incoming object is tested for a primary interface to determine which Map Class should handle the retrieval of that object.
- Generate Update Task — SmartPlant Adapter searches for an existing SmartPlant Electrical item to update. If an item is found, it generates an Update Task in the To Do List. The properties of the incoming object are compared against the properties of the existing item. When differences are found, property updates are added to the task. The list of properties to be compared is defined by the properties in the tool schema.
- Generate Create Task — If no item is found to update, a Create Task is generated in the To Do List. Property updates are added to the task for each of the incoming property values. The list of properties to be added is defined by the properties in the tool schema.
- Generate Claim Task — If no item is found in As-Built, a Claim Task is generated in the To Do List. Property updates are added to the task for each of the incoming property values. The list of properties to be added is defined by the properties in the tool schema.

- Retrieve Relationships — Depending on the class being retrieved, SmartPlant Adapter defines the types of relationships that are to be retrieved for each data object.

SmartPlant Adapter Actions in the Tombstone Container

SmartPlant Adapter iterates over all of the objects in the Tombstone Container and performs the following actions:

- Generate Delete Task — When SmartPlant Adapter encounters a delete instruction in the tombstone container, SmartPlant Adapter searches for an existing item in the SmartPlant Electrical project that is correlated to the item that was deleted in the other application. If such an item is found in the project and this item does not exist in As-Built, a Delete Task is generated in the To Do List.

- Generate Claim Task and Delete Task — When SmartPlant Adapter encounters a delete instruction in the tombstone container, SmartPlant Adapter searches for an existing item in the SmartPlant Electrical project that is correlated to the item that was deleted in the other application. If such an item is not found in the project but this item exists in As-Built, a Claim Task and Delete Task are generated in the To Do List.

- Generate Unclaim Task — When SmartPlant Adapter encounters an unclaim instruction in the tombstone container, SmartPlant Adapter searches for an existing item in the SmartPlant Electrical As-Built that is correlated to the item that was unclaimed in the other application. If such an item is found in As-Built, an Unclaim Task is generated in the To Do List.

- Remove Delete Task — When SmartPlant Adapter encounters a resurrect instruction in the tombstone container, SmartPlant Adapter searches for an existing Delete Task that is correlated to the item for which the resurrect instruction was issued. If such a task is found, this Delete Task is removed.

Related Topics

Publishing: An Overview

In an integrated SmartPlant environment, the authoring tools share data and relationships when you publish documents containing the data and relationships. The publishing process involves selecting a document to publish, assigning it to a workflow when necessary, and specifying a version and revision of the document if specified in SmartPlant Foundation. For most documents, the software also publishes the data that is associated with the document when you publish.

The authoring tools (SmartPlant P&ID or SmartPlant Instrumentation, for instance) publish data in .xml format. All leading and trailing spaces are trimmed from all strings and from all values without units. These spaces will not appear in the published data file. The software then loads the data from the .xml files to the SmartPlant Foundation database. After the data is loaded into SmartPlant Foundation, you can retrieve the data into other authoring tools.

When you publish documents, the software does the following things:

- Creates a new master document and the first revision in SmartPlant Foundation the first time that you publish a particular document. From that point on, the software creates new versions and revisions each time that you publish the document. The software relates revisions to the master document. You can publish subsequent revisions into a workflow, which can be a different workflow from the original publish action. Changes in the document status of a related revision change the status of the subsequently published versions and revisions of the document.

- Publishes a visual representation of the document that you can view without the authoring tool. For many applications, this is an Intergraph proprietary file, called a *RAD* file. The viewable file can also be an Excel workbook or another viewable file type, such as .pdf or .doc. Users can review and edit the visual representation of the document using SmartPlant Markup.

- Publishes associated data, depending on *workflow* approval. If the data is approved and loaded, it is used for reporting and subsequent retrieval by downstream applications. The software publishes only meaningful engineering data. The published data is not enough to recreate the document in the original authoring tool.

- The software publishes some document types without the associated data, such as reports from authoring tools (for example, line lists in P&ID). You can submit documents published without data to workflows just like documents with data. The document types and data that you can publish depend on the authoring tool that you use.

For more information about revisions and versions, see the *SmartPlant Foundation Web Client User's Guide*.

Reasons to Publish

You can publish documents and associated data for several reasons:

- To exchange and enhance data among tools, to avoid creating data multiple times in multiple authoring tools

- To report on common data that originates in multiple tools

- To provide enterprise-wide accessibility to published documents

- To manage change, including workflow history and document revision management

You can also publish documents to share information with users in other tools without going through a formal workflow. To share data, you can publish a document to a **For sharing** workflow that has only a load step so that the data is loaded into SmartPlant Foundation as soon as you publish the document.

You can also publish a document by not assigning the document to a workflow, but rather by using the default workflow from SmartPlant Foundation. When you do not select a workflow for a document during publishing, SmartPlant Loader loads the document into SmartPlant Foundation as soon as it reaches the top of the Loader queue.

Document Types for Publishing

Each authoring tool publishes different documents and data. For details of documents that each authoring tool can publish and retrieve, see Tool Documents That are Published and Retrieved.

The published PBS document contains information about the physical plant with a structure consisting of plants, areas, and units. The default structure is plant/area/unit, but you can define a custom hierarchy in the Schema Editor. When a PBS document is published from SmartPlant Foundation, the authoring tools are notified about the plant, areas, and units that need to be created in each authoring tool.

The project breakdown structure, project definition document, and project list contain information about projects and their statuses.

The project breakdown structure contains a single project and the hierarchy of contracts under that project in a plant/project structure. The project definition document contains information for a single project that needs to be created in the authoring tool. The project list contains a list of all projects in a plant, and it is used by those authoring tools that create all projects at one time.

 Note

- The plant breakdown structure and project breakdown structure used in the authoring tools must match the structure in SmartPlant Foundation for publishing from the authoring tools and object correlation to work correctly.

Publish Documents from SmartPlant Electrical

1. Click **SmartPlant > Publish**.

    Tips

    - This command is available only if you have registered the active plant using the SmartPlant Registration Wizard. For more information, see *SmartPlant Engineering Manager Help*.

    - If you logged onto SmartPlant Electrical with a user name that is not defined in the integrated environment, you are prompted to log on when you use this command.

    - The documents that appear in the **Selected documents** list on the **Publish** dialog box when it first opens are documents that you selected in SmartPlant P&ID before you clicked the **Publish** command.

2. Add any additional documents to the **Selected documents** list by clicking the **Engineering Tool, File System**, or **Find** buttons and choosing other documents to publish.

3. Edit information as necessary for the selected documents.

    Tips

    - When multiple documents are selected, only property values shared by all the selected documents appear in the table. Changing a value in the table changes that value for all of the selected documents.

4. Specify a method of publishing in the **Operation** box:

    - Click **Publish now** to immediately start the publishing process as soon as you click **OK**.

    - Click **Background publish** to publish the selected documents immediately as a separate process, allowing you to perform other tasks at the same time. When you use this feature, an e-mail message alerts you when the process is complete.

    - Select the **Scheduled publish** option to indicate that the publish process should be run in batch mode, if the authoring tool supports scheduled batch publishing.

5. Click **OK** to complete the publishing procedure.

📝 **Note**

- When the publish is complete, the following dialog box appears. If the **View Log** button on the dialog box is enabled, messages are available concerning the operation. These messages may include errors or warnings or even informational messages. Click the **View Log** button to see these messages.

Related Topics
- *Publishing: An Overview*, page 517

Find Documents to Publish from SmartPlant Electrical

1. Click **SmartPlant > Find Documents to Publish**.

💡 **Tips**

- This feature is also available by clicking **Find** on the **Publish** dialog box.

- This command is available only if you have registered the active plant using the SmartPlant Registration Wizard. For more information, see *SmartPlant Engineering Manager Help*.

- The **Find Documents to Publish** command determines which documents need to be published or re-published and displays the results on the **Find Documents to Publish** dialog box.

2. From the **Select documents to publish** list on the **Find Documents to Publish** dialog box, check the box beside the documents that you want to publish.

💡 **Tip**

- You can quickly select the entire list by clicking **Select All**, or you can clear the entire list by clicking **Clear All**.

3. Click **OK** to accept the selections. The documents you selected to publish now appear in the **Documents to Publish** list on the **Publish** dialog box, and are ready to be published. For more information about publishing documents, see *Publish Documents from SmartPlant Electrical*, page 519.

📝 **Note**

- The lists displayed on the **Find Documents to Publish** dialog box are compiled at the time indicated in the **Last search performed** box. You can update the lists by clicking **Update**, but this process can be time-consuming, depending on whether you are running the applications in *synchronous* or *asynchronous* mode.

Related Topics
- *Publishing: An Overview*, page 517

Retrieving: An Overview

When you retrieve documents into an authoring tool, you are retrieving the document data that was published by another authoring tool. For example, in SmartPlant Instrumentation, you can retrieve engineering information from a published P&ID into the SmartPlant Instrumentation database.

The authoring tools provide commands that let you select a document and retrieve it into that tool. You can use either the **SmartPlant > Retrieve** command to open a wizard that assists you in retrieving applicable documents, or with some authoring tools, you can configure an automatic retrieval feature.

Note

- The software trims all leading and trailing spaces from all strings and from all values without units of measure. These spaces do not appear in the retrieved data file.

Additionally, you can access the Web Client through the **SmartPlant** > **Browser** command. This allows you to select the document or documents that you want to retrieve from your Web Client To Do List, the tree view, or by using the Web Client search functionality. After you select the documents that you want to retrieve, you can use the **Retrieve** command on the Web Client **SmartPlant** menu to start the retrieval process.

The **Retrieve** command provided in the authoring tools is slightly different from the **Retrieve** command available in the SmartPlant Foundation Web Client. The Web Client presents a list of documents from which you can select those you want to retrieve. However, when you use the command from an authoring tool without first selecting documents, the software searches the SmartPlant Foundation *project* for documents to retrieve, and these are presented in a list on the **Retrieve** dialog box.

You can retrieve a document in two ways:

- **As published** - Retrieves only the data the authoring tool originally published with the selected revision and version of the document. Retrieving as-published data retrieves the .XML file the authoring tool published from the appropriate SmartPlant Foundation vault.

- **With the latest data** - Retrieves the latest data associated with the selected document in the SmartPlant Foundation database. If another, more-recently published document contains updates to objects in the selected document, the software retrieves the most current data in the SmartPlant Foundation database for those shared objects. When you retrieve the latest data, SmartPlant Foundation generates an .XML file containing the published data.

Document Types for Retrieval

The types of documents that you can retrieve depend on the authoring tool that you are using. For details of documents that each authoring tool can publish and retrieve, see Tool Documents That Are Published and Retrieved.

From the authoring tools, you can retrieve the plant breakdown structure (PBS) and project documents. The PBS and project documents, created in SmartPlant Foundation and published, are retrieved by authoring tools to provide information about the plants, areas, units, projects, and contracts that need to be created in the authoring tool so that the information is consistent across all authoring tools.

The PBS document published by SmartPlant Foundation contains information about the physical plant with a structure consisting of plants, areas, and units. The default structure is plant/area/unit, but you can define a custom hierarchy in the Schema Editor. The project breakdown structure, project list, and project definition document contain information about the project or projects and their statuses in a plant/project structure.

📝 Notes

- Retrieving the project breakdown documents and the PBS into SmartPlant Engineering Manager creates the appropriate structures automatically.

- When using SmartPlant Instrumentation, you must create the plant hierarchy according to the PBS information in SmartPlant Foundation before you retrieve either the PBS or the project definition document. You must create a plant hierarchy with at least three levels with a minimum of one unit before you can retrieve the PBS and project definition document.

Data Handling After Retrieval

The authoring tool that you use also determines how the system deals with changes in downstream data when you retrieve a document. SmartPlant P&ID, SmartPlant Instrumentation, SmartPlant Electrical, and Zyqad analyze the impact of the newly retrieved data on the existing database, then place tasks on the authoring tool's **To Do List** that allow you to create, delete, or modify items at the appropriate time in the design process. The **To Do List** gives you the opportunity to view and understand potential changes before accepting, deleting, or modifying those changes.

Design Basis

Objects that tools retrieve from other authoring tool documents can become the design basis for objects in downstream documents. Objects that become the design basis for other objects can be specific objects that get richer as they move through the lifecycle or they can be schematic or logical objects in one application that evolve into more detailed objects downstream.

Design basis is implicit based on retrieval; you do not have to define it. For example, a pump retrieved from a *PFD* becomes the design basis for a pump in the P&ID. When you change common properties for the pump and retrieve the changes into SmartPlant P&ID, tasks to update the pump automatically appear in the **To Do List**. The same process works for logical items that are a design basis for other items, such as a P&ID tag in SmartPlant P&ID can evolve into a control loop with associated tag numbers in SmartPlant Instrumentation.

Related Topics

- *Open the To Do List*, page 527

Retrieve Documents to SmartPlant Electrical

1. Click **SmartPlant > Retrieve**. The **Retrieve** dialog box appears.

   💡 **Tips**

   - This command is available only if you have registered the active plant using the SmartPlant Registration Wizard. For more information, see *SmartPlant Engineering Manager Help*.

   - If you logged onto SmartPlant Electrical with a user name that is not defined in the integrated environment, you are prompted to log on when you use this command.

   - The **Retrieve** command searches the SmartPlant Foundation project for documents that are ready to be retrieved into your tool and displays them in the **Documents to retrieve** list.

2. In the **Document type** box, specify the type of document to be retrieved.

3. In the **Show** section, select **Documents to be retrieved only** to include documents that have been retrieved previously and have been published again since the last retrieval. Select **All documents** to include all revisions and types of documents. Select **New documents only** to retrieve documents that have not been retrieved yet.

4. In the **Documents to retrieve** list, select the check box beside each document you want to retrieve. To help identify the documents, review the details in the **Type**, **Revision**, **Version**, and **Last Retrieved** columns.

Tip

- To quickly select the entire list, click **Select All**. To quickly cancel the selections, click **Clear All**.

5. For each document you checked, use the **Retrieve Option** column to specify whether you want to retrieve the document with the latest data or retrieve it as published.

6. Click **OK** to retrieve the specified documents.

Notes

- Check the **Batch retrieve** option if you want the retrieve process to run in batch mode. If you select this option, an e-mail message will alert you when the process is complete. Otherwise, the retrieval process begins when you click **OK**.

- The Deleted and Unclaimed Objects document is retrieved automatically every time you retrieve, if there is a newer version of this document since the last retrieval. The document is not included in the list, but it is retrieved automatically, when necessary, to ensure that the applicable information is updated.

- When the retrieval process is complete, the following dialog box appears. If the **View Log** button on the dialog box is enabled, messages are available concerning the operation. These messages may include errors or warnings or even informational messages. Click **View Log** to see these messages.

Related Topics

- *Retrieving: An Overview*, page 522

Using the To Do List: An Overview

The **To Do List** keeps track of tasks required to keep your plant updated with information that you retrieve. Specifically, this feature provides a list of all the items that must be added, deleted, or modified to bring the database into agreement with the newest information retrieved.

All the users of a plant database share a single **To Do List**. You can view the entire **To Do List** to see the tasks that have been completed and those that are still pending. Some types of tasks have code associated with them, and if you run those tasks, the software modifies the database.

From the **To Do List**, you can perform tasks, defer them, or delete them. Additionally, you can view properties for each task in the **To Do List**. The information that is available is specific to the type of task that is selected: **Create** tasks, **Update** tasks, and **Delete** tasks, but certain information is provided for all types.

Each task in the **To Do List** has a status. Initially, the status is **Open**. If you successfully run the task, the status changes to **Complete** ✓. If you attempt to run a task, but the process is not completed successfully, the status is changed to **Error** ⊕. **Error** status occurs, for example, if you attempt to run an **Update** task assigned to a drawing that is not currently open. You can also postpone running a task and change its status to **Deferred** ▨.

📝 **Notes**

- When an **Update** task is created in the **To Do List**, the units of measure used in properties are automatically converted to the defaults for the plant. The units of measure used for properties in **Create** tasks are converted to the plant defaults when the task is run.

- By clicking the heading of columns on the **To Do List**, you sort the list by the values in that column.

- SmartPlant Electrical does not support task dependencies. For this reason, you must run **Create** tasks in the appropriate sequence for items that have a dependency relationship. For example, if the **To Do List** contains **Create** tasks for a motor and for some related process equipment, the software cannot run the task for the motor first because in the SmartPlant Electrical database, the process equipment ID already appears as a property of the motor. You must first run the **Create** task for the process equipment, and only then for motor. In cases where the software fails to run related **Create** tasks, you can restore the tasks and run them again in the correct sequence.

Related Topics
- *Defer a Task on the To Do List*, page 529

Open the To Do List

Click **SmartPlant** > **To Do List** on the main menu bar to open the **To Do List** dialog box, from which you can organize and run tasks.

Related Topics
- *Defer a Task on the To Do List*, page 529
- *Remove a Task from the To Do List*, page 529
- *Run a Task from the To Do List*, page 528
- *Using the To Do List: An Overview*, page 525

Modify To Do List Task Properties

1. Click **SmartPlant** > **To Do List**.

2. From the task list on the **To Do List** dialog box, select the task or tasks to edit.

3. On the toolbar, click **Properties** .

   💡 **Tip**

   - You can also open the **Task Properties** dialog box by double-clicking a task in the **To Do List**.

4. On the **Task Properties** dialog box, change the properties as necessary.

   📝 **Note**

   - The properties available for each task depend on the type of task. For all types, you can edit the status, description, and notes for the task. For **Create** tasks, you can edit the catalog item being created and turn off or on the values assigned to different properties for that item. For **Update** tasks, you can turn off or on the new values assigned to properties for the item.

Related Topics
- *Using the To Do List: An Overview*, page 525

Run a Task from the To Do List

1. Click **SmartPlant** > **To Do List**.

2. On the **To Do List** dialog box, select the task or tasks to run.

   💡 **Tip**

   - You can select all the tasks on the **To Do List** by pressing **Ctrl + A**.

3. Click **Run Task** .

   Related Topics
   - *Using the To Do List: An Overview*, page 525

Defer a Task on the To Do List

1. Click **SmartPlant** > **To Do List**.

2. On the **To Do List** dialog box, select the task or tasks that you want to defer.

3. On the toolbar, click **Defer Task** .

   Related Topics
   - *Using the To Do List: An Overview*, page 525

Remove a Task from the To Do List

1. Click **SmartPlant** > **To Do List**.

2. On the **To Do List** dialog box, select the task or tasks that you want to remove.

3. On the toolbar, click **Delete Task** .

   💡 **Tip**

   - You can use the **Delete** key to delete the selected tasks.

   📝 **Note**

   - You can display deleted tasks on the **To Do List**. For more information, see *Display Deleted Tasks on the To Do List*, page 529.

   Related Topics
   - *Using the To Do List: An Overview*, page 525

Display Deleted Tasks on the To Do List

1. Click **SmartPlant** > **To Do List**.

2. On the **To Do List** dialog box toolbar, click **View Deleted Tasks** .

   💡 **Tip**

   - When you display deleted tasks, clicking **Delete Task** permanently removes the task from the database.

   - To return to the current task list, click **View Task List**.

   Related Topics
   - *Using the To Do List: An Overview*, page 525

Update the To Do List Display

- On the **To Do List** dialog box toolbar, click **Refresh** .

Related Topics
- *Using the To Do List: An Overview*, page 525

Customizing Your Data

Many of the data values that you use in SmartPlant Electrical appear in select lists. You can customize the select lists by modifying existing values, hiding or displaying particular values, or adding new values using the **Update Select Lists** dialog box. For further information on how to use this feature, refer to Online Help for the Data Dictionary Manager.

The software uses the values in certain select lists to filter the values of other properties or to perform calculations. You should proceed with extreme caution before modifying any of these select lists, because doing so could adversely affect your data. The following select lists belong to this category:

The following select lists are used in this way and are locked for changes by the user:

- Application
- Backup Type
- Boolean Values
- Cable Category
- Cable Set Type
- Cell Function
- Cell Usage
- Circuit Mode
- Circuit Type
- Conductor Arrangement
- Conformity To Standard
- Electrical Equipment Class
- Electrical Equipment Sub Class
- FLA Calculation Flag
- Frequency
- Naming Type
- Number of Phases
- Phase Arrangement
- Project Setting Type
- Rated Voltage
- Restart Flag
- Routing

- Segregation Level
- Supply AC/DC Flag
- System Cable Category
- System I/O Type
- Task Status
- Task Type

Electrical Calculations

The software calculates electrical power requirements for motors based on the load data you enter, which in turn comes from the motor manufacturer. On the **Electrical Motor Common Properties** dialog box, **Electrical Data** tab, the software displays the results of the electrical power requirement calculations.

Input Data

The software requires the following data, which you enter on the **Electrical Motor Common Properties** dialog box **Electrical Motor Common Properties** dialog box, **Load Data** tab for the calculation:

Motor rated power - Specifies the maximum mechanical power requirement for the motor.

Brake power - Specifies the mechanical power requirement for the motor at the demand point.

Demand factor - The software calculates the demand factor as the ratio of the brake power to the motor rated power. The demand factor represents the operating point of the motor that the software uses for the calculations.

Power factor - The software calculates the power factor at the operating point based on the standard power factor values at 50%, 75%, and 100% of the motor rated power.

Efficiency - The software calculates the efficiency at the operating point based on the standard efficiency values at 50%, 75%, and 100% of the motor rated power.
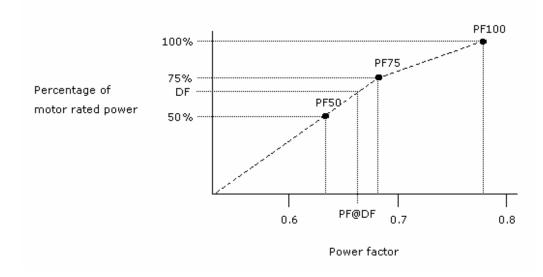
Power Factor Calculations

The software calculates the power factor and efficiency at the operating point by assuming a linear relation between the standard values at 50%, 75%, and 100% of the motor rated power and interpolating accordingly. For an operating point below 50% of the motor rated power, the software extrapolates using the known slope between the values at 50% and 75% of the motor rated power.

The following values apply to power factor calculations:

- PF50 = power factor @ 50% (number between 0 and 1)
- PF75 = power factor @ 75% (number between 0 and 1)
- PF100 = power factor @ 100% (number between 0 and 1)
- PF@DF = power factor @ demand factor (number between 0 and 1)
- DF = demand factor (number between 0 and 1)
- EF@OP = efficiency @ operating point (number between 0 and 1)

The graph shows how the software uses the user input values to calculate the power factor at a given demand factor.

Based on the graph, the software calculates the power factor for different ranges of the demand factor using the following formulas.

For a demand factor between 75% and 100%:

$$PF@DF = PF75 + \left[\frac{(PF100 - PF75) * (DF - 0.75)}{0.25} \right]$$

For a demand factor between 50% and 75%:

$$PF@DF = PF50 + \left[\frac{(PF75 - PF50) * (DF - 0.5)}{0.25} \right]$$

For a demand factor below 50%:

$$PF@DF = PF50 - \left[\frac{(PF75 - PF50) * (0.5 - DF)}{0.25} \right]$$

The software uses similar set of formulas to calculate the efficiency.

Electrical Power Calculations

The software calculates the rated, particular, and consumed load, that is, the electrical power, using the following formulas (all values refer to consumed loads at the operating point):

$$\text{Electrical active load (kW)} = \frac{\text{Mechanical power (kW)}}{\text{EF@OP}}$$

$$\text{Reactive load (kVAR)} = \text{Active load (kW)} * \frac{\sqrt{1 - (PF)^2}}{PF}$$

$$\text{Apparent load (kVA)} = \sqrt{(\text{Active load (kW)})^2 + (\text{Reactive load (kVAR)})^2}$$

📝 Notes

- For static loads, the software uses the values of the consumed and rated electrical power to calculate the electrical quantities. The software supports a value of efficiency for non-mechanical systems.

- For capacitors and harmonic filters, the active load property is disabled and has a null value.

- For heaters, the reactive load property is disabled and has a null value.

Calculation of Full Load Current

The software uses the following formulas to calculate full load current.

Loads

$$\text{Full load current (A)} = \frac{\text{Rated VA}}{k * \text{Rated voltage (V)}}$$

Battery Banks, Generators, Converting Equipment, Instruments, and Cabinets

$$\text{Full load current (A)} = \frac{\text{Rated W or VA or VAR}}{k * \text{Rated voltage (V)}}$$

Where:

- k=1 for DC and single phase AC
- k=√
- Rated voltage is the line voltage

Calculation of Synchronous Speed

The software calculates the synchronous speed of a motor or a generator according to the formula:

$$\text{Synchronous speed (rpm)} = \frac{\text{Frequency (Hz)} * 60}{0.5 * \text{Number of poles}}$$

Related Topics
- *Calculate Power Requirements for an Electrical Motor*, page 130

Earth Loop Impedance Calculation

Earth loop impedance is the total resistance between the current source and earth when clearing a faulty current to earth. The earth loop impedance determines the maximum potential between the point at which the fault occurs and earth. The software performs earth loop impedance per cable and calculates the total impedance as the sum of the grounding path impedance, external loop impedance, and impedance of the cable carrying the current. According to the user's requirement, the grounding path can be through the cable armor, or through a separate grounding cable.

For details of the formula used for the earth loop impedance calculation, see the Cable Algorithms.xls Excel file.

(If the **Cable Algorithms.xls** file does not display when you click the link, you can open the file manually from the folder where the SmartPlant Electrical program is installed.)

Related Topics
- *Earth Loop Impedance Tab (Cable Common Properties Dialog Box),* page 607

Required Data for Cable Sizing

The software performs cable sizing via a number of stages: thermal sizing, voltage drop calculation at normal running, and for a motor, voltage drop calculation at starting.

You must enter all the data required for the thermal sizing and voltage drop stages in order for the software to perform sizing. In addition, you can specify optionally for the software to perform short circuit sizing. For further details of the data properties, see the **Cable Common Properties** dialog box and refer to the appropriate dialog box tab.

For details of the formula used for the algorithms, see the Cable Algorithms.xls Excel file.

(If the **Cable Algorithms.xls** file does not display when you click the link, you can open the file manually from the folder where the SmartPlant Electrical program is installed.)

📝 Note

- The software performs sizing by selecting a matching reference cable from the **Reference Data Explorer**, so only those reference cables that you created are available for selection. You should therefore ensure that you create in the **Reference Data Explorer** all possible cable types and sizes that you may want to use.

Thermal Sizing

In the thermal sizing step, the software attempts to locate from the **Reference Data Explorer** the smallest size reference cable of the specified type and construction, with a derated ampacity equal to or greater than the full load current of the connected load. If the software cannot find a single cable that meets this requirement, it continues with groups of identical cables in parallel, from the minimum conductor size, up to the maximum permitted number of parallel cables. You can specify the minimum conductor size, the maximum permitted number of parallel cables, and the default suffix naming method in the Options Manager.

The software requires the following data for this stage of the calculation:

Cable specification (General tab) - Specifies the family of cables within which the software looks for a suitable reference cable.

Conductor arrangement (General tab) - Specifies the desired conductor arrangement of the reference cable.

Full load current [FLA] (Sizing Data tab) - Specifies the nominal current for a load associated with the cable, or a value that you type manually.

Routing (Sizing Data tab) - Indicates whether the cable routing is in the air or under the ground, and as a result determines the appropriate basic ampacity value that the software requires for cable sizing.

Conformity to standard (Sizing Data tab) - Determines the electrical standard, IEC or NEC, that the software uses. If you select the NEC standard, the software multiplies the full load current by a factor of 1.25.

Derating factor / Utilization factor / Ambient temp. factor (Sizing Data tab) - Specifies properties that the software divides the nominal current by to obtain the derated cable ampacity.

Voltage Drop Calculations

The software calculates the voltage drop across the cable under full load conditions, based on the cable length and other factors and compares it with the maximum allowable voltage drop that you specify. The software attempts to locate a cable with a sufficiently large conductor size, based on the thermal sizing, that gives a voltage drop lower than the allowed maximum value. In addition, the software calculates the maximum cable length for which the voltage drop does not exceed the maximum allowable value. For a motor, the software also calculates the percentage voltage drop under starting conditions following the same methodology.

The software requires the following data for this stage of the calculation:

Reference cable (General tab) - The reference cable you select provides, in addition to the basic ampacity values, the values of the **Resistance per unit length** and **Reactance per unit length** properties that the software requires for the voltage drop calculation.

Estimated length / Design length (Design Data tab) - Indicates the length of the cable that the software uses to calculate the voltage drop. The software uses the design length if available. Otherwise, it uses the estimated length.

Supply (Sizing Data tab) - Determines the formula that the software uses for the voltage drop calculation, depending on whether the supply is alternating current (AC) or direct current (DC).

Phases (Sizing Data tab) - Determines the formula that the software uses for the voltage drop calculation, depending on whether an AC supply is single- or three-phase.

Load voltage (Sizing Data tab) - Specifies the nominal voltage for a load associated with the cable, or a value that you type manually.

Starting current (Sizing Data tab) - Specifies the starting current for a load or any other electrical item associated with the cable, or a value that you type manually for a load or any other electrical item connected to a power cable.

Power factor at full load (Sizing Data tab) - Specifies the power factor under normal running conditions for a load associated with the cable, or a value that you type manually.

Power factor at starting (Sizing Data tab) - Specifies the power factor under starting conditions for a motor associated with the cable, or a value that you type manually for a motor.

Allowable voltage drop at full load (Sizing Data tab) - Specifies the maximum allowed voltage drop under normal running conditions for the load. To ignore the voltage drop, type a value of 100% for this property.

Allowable voltage drop at starting (Sizing Data tab) - Specifies the maximum allowed voltage drop under starting conditions for a motor. To ignore the voltage drop, type a value of 100% for this property.

Short Circuit Sizing

You can specify that you want the software to perform short circuit sizing by selecting the **Use short circuit cable sizing** check box on the **Sizing Data** tab. The software examines each reference cable that already meets the sizing requirements for the thermal and voltage drop criteria. Based on the short circuit parameters and conductor short circuit coefficients, the software calculates the minimum required conductor cross-section area for the cable in cmils and displays the value in the **Required area (cmils)** box. The algorithm compares the results of this calculation with the value in the **Conductor Area cmils** lookup table for the reference cable in question to determine whether the reference cable meets the short circuit sizing criteria. The software checks each reference cable in turn until it finds the smallest size cable that meets the short circuit sizing criteria. It is important that the lookup tables contain suitable values to enable the software to perform the calculation successfully.

The software requires the following data for this stage of the calculation:

Material (General tab) - Select the desired material for the current-carrying conductors.

Insulation (General tab) - Select the desired material used for insulating the current-carrying conductors.

Use short circuit cable sizing (Sizing Data tab) - You must select this check box to enable the software to perform the short circuit sizing calculation.

Duration (Sizing Data tab) - Type the maximum time for which the cable is expected to withstand a fault short circuit.

Fault current (Sizing Data tab) - Type the maximum current that the cable is expected to withstand in the event of a short circuit fault current.

📝 **Note**

- The software displays in the **Calculation information** box the message '**Not found**' under the following conditions: when no reference cable exists corresponding to the cable specification or arrangement, when there are missing values for ampacity, resistance, or reactance, or when the number of parallel cables exceeds the specified limit in the Options Manager.

Related Topics
- *Batch Size Cables Window*, page 594
- *Batch Size Cables*, page 186
- *Size a Cable*, page 184
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Custom Validations and Calculations: An Overview

The software uses validations to check the input values of properties and if required, to change the values of other properties prior to processing by the SmartPlant Electrical business layer.

Calculations provide a means of allowing data input for a property using a separate dialog box. When you specify a calculation, an ellipsis appears beside the value of the property in the **Properties** window of SmartPlant Electrical or beside the appropriate text box on a **Common Properties** dialog box.

Related Topics
- *Implement Calculation*, page 543
- *Implement Validation*, page 543

Implement Validation

1. Create a VB project with type ActiveX DLL.

2. On the project **Properties** dialog box, type a project name.

3. Ensure that the SPELCustomInf.tlb file is registered on the machine where you are going to build your custom .dll file. This file should be located in the folder where SmartPlant Electrical is installed and contains the appropriate interfaces that the VB project must reference.

    💡 **Tip**

    - Validation implements the interface ISPELValidate, which contains the DoValidate method and LastErrorMessage property.

4. In the **References** dialog box, ensure that **SP Electrical Customization Interfaces** is selected, then click **OK**.

5. Create a reference to the file C:\Program Files\Common Files\Intergraph\Rad\ISPClientData3V2.tlb.

6. Create a class containing your code (for an example of a validation task, see *Validation Code Example*, page 545).

7. Compile the .dll file, place it in the desired folder, and register it in the Registry of the machine on which you are running SmartPlant Electrical by entering the following code at the command prompt:

    ```
    Regsvr32 <File name>.dll
    ```

💡 **Tips**

- It is recommended that you compile the .dll file on a machine where you are going to run SmartPlant Electrical, to facilitate testing of the code. In this case, you do not need to register the .dll file.

- If it is not possible to compile the .dll file on a machine where SmartPlant Electrical runs, make sure that you register the ISPClientData3V2.tlb and SPELCustomInf.tlb library files on the machine where you perform the compilation.

8. Open the Data Dictionary Manager (Data Dictionary Manager.exe) and click **Database Tables** ⊞.

9. Select the desired entity (for example, **Plant Item**) and double-click the required property of that entity (for example, **Name**).

💡 **Tip**

- You need to have an understanding of the database table structure in order to decide which entity and property to select.

10. Beside **Validation ID**, type a ProgID which is usually made up of <VB Project name>.<Class name>. For example, if the VB Project name is MyCustomValidation and the class name is Class1.cls, the ProgID you would type is MyCustomValidation.Class1.

💡 **Tip**

- You can associate each item type property with one class only; however, the function code in the class can specify any number of cases or conditions.

11. Run SmartPlant Electrical to check that the code works (make sure that you are connected to the correct plant).

📝 **Notes**

- The software performs the custom validation whenever you change the value of the property in the **Properties** window, in the Tabular Editor, and if appropriate, in a **Common Properties** dialog box. The software does not validate changed data input that originates from elsewhere, for example, lookup tables.

- The software performs custom validations prior to performing validations in the business object layer; however, if a custom validation conflicts with a business layer rule, the software ignores the custom validation.

- If a custom validation affects other properties that have their own custom validation definitions, the software only performs the validation for the property where you change the value directly; it ignores the custom validations for the other properties.

Implement Calculation

1. Create a VB project with type ActiveX DLL.

2. On the project **Properties** dialog box, type a project name.

3. Ensure that the SPELCustomInf.tlb file is registered on the machine where you are going to build your custom .dll file. This file should be located in the folder where SmartPlant Electrical is installed and contains the appropriate interfaces that the VB project must reference.

    💡 **Tip**

    -   Calculation implements the interface ISPELCalculate, which contains the DoCalculate method.

4. In the **References** dialog box, ensure that **SP Electrical Customization Interfaces** is selected, then click **OK**.

5. Create a reference to the file C:\Program Files\Common Files\Intergraph\Rad\ISPClientData3V2.tlb.

6. Create a class containing your code.

7. Compile the .dll file, place it in the desired folder, and register it in the Registry of the machine on which you are running SmartPlant Electrical by entering the following code at the command prompt:

    ```
    Regsvr32 <File name>.dll
    ```

    💡 **Tips**

    -   It is recommended that you compile the .dll file on a machine where you are going to run SmartPlant Electrical, to facilitate testing of the code. In this case, you do not need to register the .dll file.

    -   If it is not possible to compile the .dll file on a machine where SmartPlant Electrical runs, make sure that you register the ISPClientData3V2.tlb and SPELCustomInf.tlb library files on the machine where you perform the compilation.

8. Open the Data Dictionary Manager (Data Dictionary Manager.exe) and click **Database Tables** ▦.

9. Select the desired entity (for example, **Plant Item**) and double-click the required property of that entity (for example, **Name**).

    💡 **Tip**

    -   You need to have an understanding of the database table structure in order to decide which entity and property to select.

10. Beside **Calculation ID**, type a ProgID which is usually made up of <VB Project name>.<Class name>. For example, if the VB Project name is MyCustomValidation and the class name is Class1.cls, the ProgID you would type is MyCustomValidation.Class1.

11. Run SmartPlant Electrical to check that the code works (make sure that you are connected to the correct plant).

Related Topics
- *Custom Validations and Calculations: An Overview*, page 541

Validation Code Example

The following code for a validation allows you to change the value for the **Name** property of a load that is a motor, but not for other item types. Furthermore, in the case of a motor, it modifies value for the **Description** property to a string made up of the item type (motor), the item tag, and the string 'custom validated'.

1. In the **General Declarations** section of the class add the following lines:

```
Option ExplicitImplements SPELCustomInf.ISPELValidatePrivate
m_strLastErrorMessage As String
```

2. Implement the DoValidate method as follows:

```
Private Function ISPELValidate_DoValidate(ByVal _ objISPItem As
ISPClientData3.ISPItem, ByVal _ strPropName As String, ByVal
varNewValue As Variant) _As Boolean    If objISPItem.ItemType.Name
= "Motor" Then        objISPItem.Attributes("Description").Value =
_       objISPItem.ItemType.Name + " " + _
CStr(objISPItem.Attributes("ItemTag").Value) + _        custom
validated"        ISPELValidate_DoValidate = True    Else
m_strLastErrorMessage = "Motors only allowed!"
ISPELValidate_DoValidate = False    End IfEnd Function
```

Related Topics
- *Implement Calculation*, page 543
- *Implement Validation*, page 543

Commands and Controls

New > Item Command (File Menu)

Creates a new item or adds a node under the selected node in the **Reference Data Explorer** or **Electrical Index**.

Note

- Clicking on the down-arrow of the **File** menu button above accesses a list of the associated commands. Clicking on the **File** menu button itself repeats the last command you used from the **File** menu.

New > SLD Command (File Menu)

This menu option is available when you select an electrical item in the **Electrical Engineer**.

Opens the **Single Line Diagram Options** dialog box, which allows you to define generation settings for a single line diagram.

New > Template Command (File Menu)

Creates a new blank template in the current plant.

Caution

- Templates created at the user-level instead of the administrator-level are not available to other users for creating drawings.

Related Topics
- *Open > Template Command (File Menu)*, page 548
- *Working with Drawing Templates: An Overview*, page 264

New > Custom Folder (File Menu)

This menu option is available when you select an equipment folder in the **Reference Data Explorer** or **Electrical Index**.

Opens the **Create a Custom Folder** dialog box, which allows you to create a custom folder within the selected equipment folder.

Open > Plant Group Command (File Menu)

Displays the **Open Plant Group** dialog box, which allows you to navigate to a specific plant group within a plant.

Related Topics
- *Connect to a Plant Group*, page 33

Open > Drawing Command (File Menu)

This menu option is available when you select a schematic drawing in the **Electrical Index**.

Opens the schematic drawing that you select.

Open > PDB Layout Report Command (File Menu)

This menu option is available when you select a PDB in the **Electrical Index**.

Opens a saved report for the power distribution board that you selected. The report opens in the format of an Excel spreadsheet.

Open > SLD Command (File Menu)

This menu option is available when you:

- Select an existing SLD drawing in the **Electrical Index**.
- Select an electrical item in the **Electrical Engineer**.

Opens a saved single line diagram for the item that you select.

Open > Template Command (File Menu)

Opens a template within the current plant. Templates are in the .spe file format. You can edit the template by changing the page setup or modifying the border and title block.

Related Topics
- *New > Template Command (File Menu),* page 547
- *Working with Drawing Templates: An Overview*, page 264

Open > Log Files Command (File Menu)

Opens a dialog box with a list of categories from which you can select and view log files.

Close Command (File Menu)

Closes the active drawing or Tabular Editor. The software saves your changes automatically.

Save Command (File Menu)

Stores the active drawing or template. If you have not already named the drawing or template, the **Save As** dialog box appears. If you have already named the drawing or template, no dialog box appears. Project location and paths are set in SmartPlant Engineering Manager and Options Manager.

Save As Command (File Menu)

Opens the **Save As** dialog box. Saves the active document to a new format.

Preferences Command (File Menu)

Opens the **Preferences** dialog box, where you can set preferences for general and single line diagram generation options, load summary report options, and delete load options.

Sheet Setup Command (File Menu)

Defines the properties of the *working sheet*. You can display and modify the following properties: the name, size, and scale used in the working sheet; and the properties of the working sheet margin. You can also select the *background sheet* you want to use.

You can save settings only for the active working sheet. These settings can also be saved so that they are used when you add a new working sheet to the document. To display a working sheet, use **Working Sheets**.

💡 **Tip**

- You can use **Tools > Customize** to place the **Sheet Setup** button on a toolbar.

Print Command (File Menu)

Sends a copy of the active drawing to a specified plotter, printer, or file. Options are available for defining the printing area, range, number of copies, and other printing characteristics. Selecting **File > Print** opens the **Print** dialog box.

Before using this command, you must install and select a printer. For help on installing a printer, see the Windows documentation.

Properties Command (File Menu)

Accesses general information about the current drawing, and opens the **File Properties** dialog box. You can review and edit the drawing summary and statistics.

Exit Command (File Menu)

Closes the active drawing and quits the software. The software checks for unsaved changes when you quit.

📝 Note

- On exiting the software, the positions and sizes of all dialog bars (**Reference Data Explorer**, **Electrical Index**, and **Electrical Engineer**), as well as any filter settings, are retained. The next time that you log on to the program, you see the same set of data displayed as it was in the previous session.

Undo Command (Edit Menu)

Reverses an action. You can change the number of actions that can be undone using **Options**. You then click the **General** tab on the **Options** dialog box.

💡 Tip

- You can use **Tools > Customize** to place the **Undo** button on a toolbar.

Related Topics
- *Options Command (Tools Menu)*, page 567

Redo Command (Edit Menu)

Repeats the most recent action taken or reverses the most recent **Undo**.

💡 Tip

- You can use **Tools > Customize** to place the **Redo** button on a toolbar.

Cut Command (Edit Menu)

Cuts selected items from the drawing and pastes them to the Clipboard. The selected items replace the previous contents of the Clipboard.

Copy Command (Edit Menu)

Copies selected items and their associated properties in the database to the Clipboard. This command replaces the previous contents of the Clipboard with the new contents.

Paste Command (Edit Menu)

Inserts the Clipboard contents at the same location the items occupied in the source document or drawing. The command is not available if the Clipboard is empty.

Delete Command (Edit Menu)

Removes the selected item from the database.

Select All Command (Edit Menu)

Selects all visible items in a view.

Note

- You can access this command when you right-click a blank area in a table. On the shortcut menu click **Select All**.

Related Topics
- *Select an Item*, page 366
- *Selecting Drawing Items: An Overview*, page 365

Duplicate Command (Edit Menu)

This command is available when you select an item in the **Electrical Index** or the **Reference Data Explorer**. When you duplicate an item, the software creates the new item with a unique item tag. If a *naming convention* is defined for the item type, the new item tag follows that naming convention.

Notes

- On the **Preferences** dialog box, **General** tab, under **Open after creating or duplicating an item**, you can specify whether to open the **Item Tag** dialog box when you duplicate the item.

- Some items have a **Common Properties** dialog box from which you can edit the most common properties for the item. You can specify in the preferences to open this dialog box directly when you duplicate the item.

Rename Command (Edit Menu)

This menu option is available when you select an item in the **Electrical Index**, **Electrical Engineer**, or **Reference Data Explorer**.

Opens the **Item Tag** dialog box, which allows you to rename the selected item.

Move Command (Edit Menu)

Moves items from one location to another. You can specify the locations by clicking the drawing sheet or by entering relative values in the Move ribbon. You can move one or multiple items at a time, and you can move select sets. You can access this command from the **Edit** menu or the main toolbar.

Note

- You can use other view manipulation commands, such as **Zoom**, **Fit**, and **Pan**, while you are using the **Move** command. When you finish manipulating the view, the software returns you to the **Move** command at the point where you were when you started manipulating the view.

Rotate Command (Edit Menu)

Rotates one or more items a precise distance or angle about a specified point. You can access this command from the **Edit** menu and the main toolbar, too.

📝 **Note**

- You can rotate an item about its center with the rotate component handle. The rotate handle appears as a green circle to the right of the selected item.

Mirror Command (Edit Menu)

Reflects one or more selected items about a line or axis that you define. You can mirror without copying, or mirror and copy. You can access this command from the **Edit** menu or the main toolbar.

📝 **Note**

- You can use mirror handles to mirror an item about its center. The mirror handle appears as a green plus sign at the right center of the item when you select the item.

Activate Connection Mode Command (Edit Menu)

Allows you to add connectors between all available connection points on symbols.

Disconnect (Edit Menu)

This menu option is available when working in a cable block diagrams and wiring drawings.

Allows you to select a cable and disconnect the wiring items that the cable is connecting. This action removes the cable from the current drawing. This action does not delete anything from the database. Note that this action only disconnects the items from each other. The software retains the association between the item and the cable.

Remove (Edit Menu)

This menu option is available when working with cable block diagrams, single line diagrams and wiring drawings.

Allows you to remove an item from a cable block diagram, single line diagram, or a wiring drawing. This action does not delete the item from the database.

Note

- This procedure only works for SLDs created using a blank SLD template. It can not be used on SLDs that were automatically created by the software.

Toggle Redlining Command (Edit Menu)

Turns on and off the redlining mode for single line diagrams.

Insert > Image Command (Edit Menu)

This command is available after opening a single line diagram (SLD) or schematic drawing.

Opens a dialog box where you can navigate to an image to insert in the drawing.

Insert > Symbol Command (Edit Menu)

This command is available after opening a single line diagram (SLD) or schematic drawing.

Opens a dialog box where you can navigate to a symbol to insert in the drawing.

Insert > Text Command (Edit Menu)

This menu option is available after opening an SLD or a schematic drawing.

Opens the **Text Properties** dialog box, which enables you to add or modify annotation or redlining text and create a watermark.

Insert > Legend Command (Edit Menu)

This menu option is available after opening an SLD drawing.

Opens the **SLD Legend** dialog box, which enables you to add a legend showing the meaning of all the symbols that appear in the default symbol folder and any other symbols that are specific to the current single line diagram.

Insert > Line Command (Edit Menu)

This command is available after opening a single line diagram (SLD) or schematic drawing.

Allows you to insert a redlining line in the drawing.

Insert > Circle Command (Edit Menu)

This command is available after opening a single line diagram (SLD) or schematic drawing.

Allows you to insert a redlining circle in the drawing.

Insert > Rectangle Command (Edit Menu)

This command is available after opening a single line diagram (SLD) or schematic drawing.

Allows you to insert a redlining rectangle in the drawing.

Text Command (Edit Menu)

This menu option is available after opening an SLD or a schematic drawing.

Opens the **Text Properties** dialog box, which enables you to modify an annotation text string that you have selected in a drawing.

Document Properties Command (Edit Menu)

This menu option is available when you open a drawing or select a drawing in the **Electrical Index**.

Opens the **Document Properties** dialog box, which enables you to enter or modify the document name, description and revisions.

Notes

- If you choose this command without first opening a pertinent document, all the options in the dialog box options appear in read-only mode.

- If you click outside the drawing, this command is not available. To access this command, first select an item in the drawing.

Common Properties Command (Edit Menu)

Opens the **Common Properties** dialog box for the selected item where you can enter electrical properties as you require. This command is only available for items that have a dialog box, such as motors, loads, converting equipment, cables, control stations, and so forth.

Table Properties Command (Edit Menu)

Opens the **Table Properties** dialog box, which enables you to specify the item type (such as **Battery Bank**, **Cable**, and **Load**), a layout for the item type which determines how the software presents the item type data, and if required, a filter.

Transformer Connections and Tapping Command (Edit Menu)

This menu option is available when you select a transformer in the **Electrical Index**.

Opens the **Transformer Connections and Tapping** dialog box, which enables you to set options for defining the types of connection (Delta, Wye, and so forth) and neutral grounding for 3-phase transformers in your project. Also allows you to specify tapping arrangements on the primary and secondary transformer windings for transformers with all types of feeds.

Previous Command (View Menu)

Restores the previous view. For example, if you zoom in and click **View > Previous**, the view changes to the view present before you had zoomed-in. This command is available only when a drawing view is active, and SmartPlant Electrical does not remember more than one view.

Refresh Command (View Menu)

Refreshes the drawing or report with updated data.

Show Only Command (View Menu)

Filters the display of the **Electrical Engineer** window and shows only the selected branches of the network structure.

To display all the existing data in the **Electrical Engineer**, close the **Electrical Engineer** and open a new session by clicking on the main toolbar.

Show in New Window Command (View Menu)

Filters the display of the **Electrical Engineer** window and shows only the selected branches of the network structure in a new **Electrical Engineer** window.

Show Items of All Plant Groups Command (View Menu)

Toggles the display of the **Electrical Index**, **Reference Data Explorer** and Tabular Editor to show all items in the plant for which you have the appropriate access rights or to show only items that belong to plant groups in the current login path. For example, in a plant hierarchy 7 with the following tree:

```
Plant 1
  • Area 100
      • Unit 100
      • Unit 101
  • Area 200
      • Unit 200
      • Unit 201
```

Logging into Unit 101 filters out all items except for those associated with Unit 101, Area 100, and Plant 1. The software filters out items associated with Unit 100 or with Area 200 and its units (Unit 200, Unit 201).

Show Related Items (View Menu)

Opens a new frame and shows all items related to main selected item.

Add Filter Command (View Menu)

Adds a filter tab to the drawings in the **Design** window. This command displays the **Select Filter** dialog box, which contains a subset of the commands in Filter Manager.

Save Settings Command (View Menu)

Stores all filters if the active view is a drawing view. Stores the table layout and column widths if the active view is a table view.

📝 **Note**

- Your settings are automatically saved.

Zoom Area Command (View Menu)

Enlarges the display of items in the active window. This command is available only when a drawing view is active.

Zoom In Command (View Menu)

Enlarges the display of items around a specified point in the active window. This command is available only when a drawing view is active.

Zoom Out Command (View Menu)

Reduces the display of items around a specified point in the active window. This command is available only when a drawing view is active.

Fit Command (View Menu)

Fits all visible items in the active view. This command is available only when a drawing view is active.

Pan Command (View Menu)

Allows you to move the display in any direction from a specific point in a drawing to see other areas of the drawing by dragging the pointer across the view. This command is available only when a drawing view is active.

Show Grid Command (View Menu)

Displays a grid so that you can place items with precision. The grid is not considered part of the drawing and does not print.

Related Topics
- *Snap Grid Command (View Menu)*, page 561

Snap Grid Command (View Menu)

Aligns items with the grid. The grid is a set of lines in the drawing background that helps you align items. When you set the **Snap grid** option, items always align with the grid lines or nearest intersection of the grid lines. Grid lines do not print.

📝 **Note**

- You can change the grid display by setting options on the **Grid** tab of the **View Properties** dialog box, accessed by clicking **View > Properties**. To view grid lines at a finer level, set the **Style** to Dynamic and adjust the grid line width with the **Density** control.

Related Topics
- *Show Grid Command (View Menu)*, page 560

Display > Properties Window Command (View Menu)

Turns the display of the **Properties** window on or off.

Toolbars Command (View Menu)

Opens the **Toolbars** dialog box, which allows you to create new toolbars and displays or hide selected toolbars. You can change toolbar color schemes and button sizes.

Rule Inconsistency Statistics Command (View Menu)

Opens the **Rule Inconsistency Statistics** dialog box, from which you can view a summary of selected items and whether they have violated any rules defined in the **Rule Manager**.

Design PDB Layout Command (Actions Menu)

This menu option is available after selecting a power distribution board in the **Electrical Index**.

Allows you to design a PDB layout for a selected power distribution board.

Batch Load Association Command (Actions Menu)

Opens the **Batch Load Association** dialog box, which allows you to associate multiple loads with a power distribution board (PDB) and display the aggregate calculated electrical consumption.

Total Bus Load Validation Command (Actions Menu)

Opens the **Total Bus Load Validation** dialog box, which allows you to validate the total loads on buses for selected power distribution boards or buses.

Generate SLD for PDB Command (Actions Menu)

This menu option is available after selecting a power distribution board in the **Electrical Index**.

Opens the **Single Line Diagram Options** dialog box, which allows you to define SLD generation settings for the selected power distribution board.

Generate Schematic Command (Actions Menu)

This menu option is available after selecting a load in the **Electrical Index**.

Opens the **Generate Schematic** dialog box, which enables you to generate a schematic drawing for the selected load.

Define Document Reference Command (Actions Menu)

Opens the **Document Reference Definition** dialog box, which enables you to define a reference for an external document.

Associate Documents Command (Actions Menu)

Opens the **Associate Documents** dialog box, which enables you to associate external documents with selected electrical items.

Global Revisions Command (Actions Menu)

Opens the **Global Revisions** dialog box, which allows you to assign and manage revisions for a group of documents.

Compare Documents Command (Actions Menu)

Opens the **Document Properties** dialog box where you can select an archived version of the current document with which you want compare the document that you have selected.

Associate Custom Symbols Command

Opens the **Associate Custom Symbols** dialog box where you can associate a customized symbol with a document type and specific item tag. You also remove a customized symbol from here.

Dissociate Command (Actions Menu)

This menu option is available when you select an electrical item in the **Electrical Engineer**.

Cancels the electrical association of the selected electrical item with its feeding item in the **Electrical Engineer**.

Move Items Command (Actions Menu)

Displays the **Move Items to Plant Group** dialog box, which allows you to select the plant group to which you want to move selected electrical items.

Related Topics
- *Connect to a Plant Group*, page 33

Cables > Batch Size Cables Command (Actions Menu)

This menu option is available when you select a cable in the **Electrical Index**.

Opens the **Batch Size Cables** dialog box, which enables you to size cables in batch mode.

Cables > Replace Cables Command (Actions Menu)

This menu option is available when you select a cable in the **Electrical Index**.

Opens the **Replace Cables** dialog box, which enables you to replace the characteristics of a group of project cables using specified properties from a common reference cable.

Cables > Apply Reference Data to Cables Command (Actions Menu)

Opens the **Apply Reference Data to Project Cables** dialog box, which allows you to select project cables for which the reference data has changed and to apply the new reference data values to those project cables.

Cables > Assign Drums to Cables Command (Actions Menu)

Opens the **Assign Drums to Cables Wizard**, which allows you to assign cables automatically to cable drums.

Cables > Predefined Routes Command (Actions Menu)

Opens the *Predefined Routes Dialog Box,* page 638, which enables you to organize cableway segments in predefined routing paths.

Cables > Batch Cable Routing Command (Actions Menu)

This menu option is available when you select a cable in the **Electrical Index**.

Opens the *Batch Cable Routing Dialog Box*, page 628, which enables you to effect cable routing in batch mode.

Cables > Define Color Pattern Command (Actions Menu)

Opens the **Define Color Pattern** dialog box, which allows you to specify wire color patterns that you can apply to cables.

Cables > Refresh Load Data for Power Cables Command (Actions Menu)

After you modify parameters that affect cable sizing in one or more loads in the plant, this option refreshes the sizing data parameters for all the cables attached to those loads.

Cables > Associate Cables with Equipment Circuits Command (Actions Menu)

Associates the 'From' side of all non-power cables for control stations or instruments with the feeder circuit for the load.

Cables > Batch Cable Side and Gland Associations Command (Actions Menu)

Opens the *Batch Cable Side and Gland Associations Dialog Box*, page 593, which enables you to batch associate cable sides with an item for non-power cables and to associate cable sides with glands for cables of all categories.

Cables > Batch Cable Connection Command (Actions Menu)

Opens the **Batch Cable Connection dialog box**, which allows you to connect numerous project cables in batch mode.

Calculate Bus Loads > Selected PDBs or Buses Command (Actions Menu)

Calculates load totals for selected buses or PDBs.

Calculate Bus Loads > All Plant Buses Command (Actions Menu)

Calculates load totals for all the buses in your plant.

Copy Driver Power Absorbed (Actions Menu)

Opens the **Copy Driver Power Absorbed** dialog box, which allows you copy the value of the **Driver Power Absorbed** of process equipment items to the **Brake Power** value of the associated motors. This functionality exists for process equipment and motors that have been retrieved from SmartPlant P&ID.

Register Report Command (Actions Menu)

Opens the **Register Report Common Properties** dialog box, which allows you to save a report based on a particular template to the database for the purpose of assigning revisions.

Gap Now Command (Tools Menu)

Performs gapping on the current drawing - that is, on the entire drawing. Gapping refers to the condition that exists when two lines intersect graphically on the drawing but do not physically intersect in the plant.

Related Topics
- *Perform Line Gapping in Your Drawing Now*, page 315

Optimize Overlapping Connectors

Repositions overlapping connectors so that they do not overlap on the drawing sheet.

Apply Options Command (Tools Menu)

☑

Opens the **Apply Options** dialog box, which enables you to apply default profiles, circuits, control stations, and lookup tables, all of which you define in the **Reference Data Explorer**.

Drawing Options Command (Tools Menu)

This menu option is available after opening a single diagram drawing.

Opens the **Single Line Diagram Options** dialog box, which allows you to redefine SLD generation settings and regenerate the drawing.

Update Select Lists (Tools Menu)

Opens the **Update Select Lists** dialog box, which allows you to add or edit items on SmartPlant Electrical select lists.

Apply Naming Conventions (Tools Menu)

Applies the naming conventions as defined in the Options Manager, to the items in your plant.

Customize Command (Tools Menu)

Allows you to customize toolbars or menus to fit your workflow.

Options Command (Tools Menu)

Changes settings that control screen appearance, document location, user information, and so forth.

📝 Note

- **Tools > Options** also allows you to set options for importing MicroStation or AutoCAD documents into the software.

Publish Plant Groups Command (SmartPlant Menu)

Saves an .xml data file containing details of the plant groups and their structure in the current site server.

Publish Command (SmartPlant Menu)

Allows you to write the information in the active document or documents from the authoring tool to an .xml file.

When you use the SmartPlant Instrumentation Interface, this command opens the **Publish to File** dialog box for specify the location and names of the files used for the published data.

Find Documents to Publish Command (SmartPlant Menu)

Opens the **Find Documents to Publish** dialog box, which helps you select the documents that you want to publish and specifies those documents that have been published before, have never been published, and need to be re-published. This list is specific to the authoring tool from which you used the command.

Related Topics
- *Publishing: An Overview*, page 517

Retrieve Command (SmartPlant Menu)

Allows you to retrieve documents into the authoring tool from published .xml documents that come from SmartPlant Instrumentation.

When you use the SmartPlant Instrumentation Interface, this command opens the **Retrieve Document** dialog box that you use for selecting the document containing the source data that was published from SmartPlant Instrumentation.

Publish to External Analyzing Tool Command (SmartPlant Menu)

Allows you to save the information in a saved electrical analysis SLD from the authoring tool to an .xml file used to publish data to the ETAP or EDSA external analyzing tools.

When you use the SmartPlant Instrumentation Interface, this command opens a dialog box that allows you to specify the location and name of the file used for publishing the data.

To Do List Command (SmartPlant Menu)

Opens the **To Do List** dialog box, which lists tasks to be performed in the authoring tool, that is, SmartPlant Electrical or SmartPlant P&ID. From this dialog box, you can run, defer, or delete tasks, and you can organize your tasks.

Related Topics
- *Using the To Do List: An Overview*, page 525

Browser Command (SmartPlant Menu)

Opens a web-based user interface that allows you to interact with SmartPlant Foundation if the active plant is registered. From this interface, you can perform a number of tasks, such as browsing for documents that have been published, using the SmartPlant Foundation To Do List to complete tasks, comparing published documents with the data in your authoring tool, and subscribing to documents in order to receive notification of changes to the documents.

New Command (Reports Menu)

Displays the **New Report Template** dialog box. This dialog box provides options for defining plant-level and user-level templates. You can assign source templates, names, formats, and descriptions for your report template and define the item type upon which you base your report. You do not have to edit the report template when you first create it.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Create a Report Template Based on an Existing Template*, page 419
- *Creating Report Templates: An Overview*, page 416
- *Using the Composite Format Report*, page 425
- *Using the Fixed Format Report*, page 423

Edit Command (Reports Menu)

Opens the **Edit Report Template** dialog box. This dialog box lists all the available report templates. You can select a template and view its properties and edit it in Microsoft Excel.

Note

- You must have valid privileges to edit a report template.

Related Topics
- *Creating Report Templates: An Overview*, page 416
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Edit a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

Delete Command (Reports Menu)

Displays the **Delete Report Template** dialog box, which allows you to select and delete a report template.

Note

- You must have valid privileges, granted in SmartPlant Engineering Manager, to delete a report template.

Related Topics
- *Delete a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

Plant Reports Command

Opens the **Plant Reports** dialog box, which displays a list of all plant-level reports associated with the current plant. This list is alphabetical. The location of these report templates is defined in Options Manager. Selecting a plant report from this list and specifying the items you want to report on generates the associated report in Microsoft Excel.

This command is available on the **Reports** menu on the main toolbar and also on the **View** menu in the Tabular Editor. If you want to run a report based on a selected subset of items, open the Tabular Editor and select the set of tags for which you want to generate the report.

Note

- The Tabular Editor is the area where you can view your project or reference items in a tabular way.

My Reports Command

Opens the **My Reports** dialog box, which displays a list of all user-level reports, which may be stored on your local workstation. This list is alphabetical.

If you want to run a report on based on a selected subset of items, open the Tabular Editor and select the set of tags for which you want to generate the report.

Related Topics
- *Edit a Report Template*, page 431
- *Generate a Report*, page 387
- *Generating Reports: An Overview*, page 386

New > Table Command (Window Menu)

Creates a table view, which allows you to view and edit item data in a tabular format. The **Table Properties** dialog box appears so that you can define the items and properties for the new table view.

Related Topics
- *Define a New Tabular Editor View*, page 101

New > Electrical Index Command (Window Menu)

Opens a new **Electrical Index** window.

New > Electrical Engineer Command (Window Menu)

Opens a new **Electrical Engineer** window.

New > Reference Data Explorer Command (Window Menu)

Opens a new **Reference Data Explorer** window.

New > Reference Electrical Engineer Command (Window Menu)

Opens a new **Reference Electrical Engineer** window.

New > Project Management Command (Window Menu)

Creates a table view, which allows you to perform various operations relating to projects. Depending on whether you are in As-Built or a project, this command opens the **Project Management (As-Built)** table or the **Project Management** table (for the project) respectively.

Related Topics
- *SmartPlant Electrical Project Management: An Overview*, page 465

Cascade Command (Window Menu)

Overlaps windows diagonally across the screen.

Tile Horizontally Command (Window Menu)

Arranges windows to fit horizontally on the screen. All the windows appear at an even distance from each other on the screen.

Tile Vertically Command (Window Menu)

Arranges windows to fit vertically on the screen. All the windows appear at an even distance from each other on the screen.

Printable Guides Command (Help Menu)

Opens the **Printable Guides** Help page for accessing the printable guides as .pdf files.

SmartPlant Electrical on the Web Command (Help Menu)

Activates your World Wide Web browser and opens the SmartPlant Electrical World Wide Web page. On this page, you can access registration and support information, learning tools, and other items to help you use SmartPlant Electrical more efficiently.

SmartPlant Electrical Web Forum Command (Help Menu)

Allows you to access Intergraph's online community of SmartPlant Electrical users. In addition to facilitating communication between Intergraph personnel and other SmartPlant Electrical users, the forum also provides access to short, informative video clips that illustrate step-by-step guidelines for performing sophisticated workflows. These clips may be viewed online, or they may be downloaded to your local machine for later viewing.

About SmartPlant Electrical Command (Help Menu)

Opens the **About SmartPlant Electrical** window.

Copy Command (Edit Menu - Tabular Editor)

Copies the value in the selected row, cell, or group of cells, to the Clipboard so that it can be pasted somewhere else. This command does not remove the value from the table. Values placed on the Clipboard remain there until newer values are copied over them.

Related Topics
- *Modify the Table Properties of an Existing Tabular Editor View*, page 104
- *Review and Edit Item Properties*, page 110

Paste Command (Edit Menu - Tabular Editor)

Places the value or values on the Clipboard into the selected row, cell, or group of cells. Values do not paste successfully if you have selected a cell for which the value is strictly generated by validation or calculation, or is read-only. When pasting, you must select the same number of rows as the number of rows that were copied to the Clipboard.

Related Topics
- *Review and Edit Item Properties*, page 110

Select All Command (Edit Menu - Tabular Editor)

Selects all the cells in the active Tabular Editor view. You can use this command to create a select set and view its properties in the **Properties** window, for instance.

✎ Note

- You can access this command when you right-click an area within the Tabular Editor. On the shortcut menu click **Select All**.

Related Topics
- *Review and Edit Item Properties*, page 110

Tabular Editor Shortcut Menu

The shortcut menu included commands that are frequently used when you are working with the Tabular Editor.

Copy

Copies the value in the selected row, cell, or group of cells, to the Clipboard so that it can be pasted somewhere else. This command does not remove the value from the table. Values placed on the Clipboard remain there until newer values are copied over them.

Paste

Places the value or values on the Clipboard into the selected row, cell, or group of cells. Values do not paste successfully if you have selected a cell for which the value is strictly generated by validation or calculation, or is read-only. When pasting, you must select the same number of rows as the number of rows that were copied to the Clipboard.

Multi-Tag Paste

Pastes data copied from a single row into the selected rows. This command is only available where you select more than one row to paste into. The software validates and calculates those properties that are based on input properties. Properties such as item tags and read-only data are not copied.

Refresh

Refreshes the table with current data from the database. Note that in a multi-user environment, more than one person may be updating data. Your opened table does not refresh automatically and you therefore need to run this command periodically to display the most recent data. Where another user has made a change on an item for which you have opened the properties in the table, the software notifies you that your input will be rejected until you first refresh the data.

Delete

Deletes the selected items from the database.

Select All

Selects all the cells in the active Tabular Editor view. You can use this command to create a select set and view its properties in the **Properties** window, for instance.

Duplicate

Creates a new item identical to the selected item. If the value of the **Comply To Naming Convention** tag property for the item is set to **True**, the duplicated item tag is based on the *naming convention* for the item type.

Rename

Enables you to rename an existing tag name. Depending on the value of the **Comply To Naming Convention** tag property for the item, you can either completely override the tag or modify only that section of the tag structure that is not protected by the naming convention.

New Item

Creates a new item of the same item type as the item type selected for the active Tabular Editor view. Where selected items belong to more than one item type, for example a motor and a motor with a space heater, the software prompts you with a list from which you need to select the type of item you want to create.

Common Properties

Opens the **Common Properties** dialog box for the selected item where you can enter the required electrical properties. This command is only available for items that have a dialog box, such as motors, cables, control stations, and so forth.

Table Properties

Opens the **Table Properties** dialog box, which enables you to specify the item type (such as **Battery Bank**, **Cable**, and **Load**), a layout for the item type which determines how the software presents the item type data, and if required, a filter.

Save Settings

Stores the table layout and the order and width of the Tabular Editor columns.

Auto Filter

Allows you to filter the tags according to various values that the selected property contains. When you click the **Auto Filter** command, arrows appear at the top of each column. Click on the arrow at the top of the column that contains the data you want to filter on and choose the value from the list that appears. You can include more than one property in the auto filter of your table, creating a chained filter.

Click this command again to remove the filter.

To create more elaborate filters and displays, you need to use the **Table Properties** command and enter options on the **Table Properties** dialog box and the **Advanced Table Properties** dialog box.

Freeze Panes

Allows you to scroll through the Tabular Editor while keeping the display of specified rows and columns constant. For instance, the first column can contain the item tag, but you want to always see the item tag as you scroll to the right end of the rows to view the supplier.

Related Topics
- *Customizing Data Display in the Properties Window: An Overview*, page 111

Select Tool Command (Main Toolbar)

Changes the pointer to the arrow-shaped selection pointer so that you can select, modify, and manipulate items. The circle at the end of the pointer arrow is the *locate zone*. While the select tool is active, the **Select Tool** ribbon is displayed.

Related Topics
- *Select an Item*, page 366
- *Selecting Drawing Items: An Overview*, page 365

Select Tool Ribbon

The **Select Tool** ribbon is displayed only when nothing is selected. After you select an item, the **Select Tool** ribbon is replaced with a ribbon for editing the selected item.

Bottom Up - Specifies where individual items in a group are located as opposed to the whole group.

Inside - Specifies that items inside the fence are selected.

Overlapping - Specifies that items overlapped by the fence are selected, as well as items inside the fence.

Top Down - Specifies that groups of items are located as opposed to individual items in a group.

Note

- To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Related Topics
- *Select an Item*, page 366
- *Select Tool Command (Main Toolbar)*, page 574

SmartPlant Reports Toolbar

The **SmartPlant Reports** toolbar is displayed when you are editing or creating a report template in Microsoft Excel.

Define - Opens the **Define Report Contents** dialog box. You can add new item types to your report and select related item type and item properties to be available to include in your report template. You can associate a sort order with your selected properties and filters for selecting your primary report item type.

Map Properties - Displays a menu of all properties associated with your report. This menu is populated with the items that you choose with the **Define** command. The

Map Properties menu contains a subset of all the available properties. You select a cell in your report template and then assign a property from the **Map Properties** list. The software places the corresponding property in the selected cell. You do not have to map all the properties in the **Map Properties** menu, and you can add properties to the menu by using the **Define** command again.

Options - Opens the **Options** dialog box, which provides options for defining header and spacing details in your report template. The **Options** button on the **SmartPlant Reports** toolbar is not available for editing a fixed format report template because you are free to place headers and data anywhere you want to on your worksheet.

✎ **Note**

- If the SmartPlant **Reports toolbar** is missing when editing report templates in Microsoft Excel, then from the Excel menu bar, click **Tools > Macro > Visual Basic Editor** and follow the steps below:

1. Select **View** > **Immediate Window**.

2. Type Sheet1.Application.CommandBars("SmartPlant Reports").Delete in the **Immediate Window**.

3. Quit Excel, and the toolbar will be displayed the next time a report is edited.

Related Topics
- *Define Command (SmartPlant Reports Toolbar)*, page 577
- *Define the Contents of Your Report Template*, page 428
- *Edit a Report Template*, page 431
- *Map Properties Command (SmartPlant Reports Toolbar)*, page 579
- *Options Command (SmartPlant Reports Toolbar)*, page 578

Define Command (SmartPlant Reports Toolbar)

Opens the **Define Report Contents** dialog box. You can add new item types to your report and select related item type and item properties to be available to include in your report template. You can associate a sort order with your selected properties and filters for selecting your primary report item type.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Edit a Report Template*, page 431

Options Command (SmartPlant Reports Toolbar)

Opens the **Options** dialog box, which provides options for defining header and spacing details in your report template. The **Options** button on the **SmartPlant Reports** toolbar is not available for editing a fixed format report template because you are free to place headers and data anywhere you want to on your worksheet.

Related Topics
- *Define the Contents of Your Report Template*, page 428
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Edit a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

Map Properties Command (SmartPlant Reports Toolbar)

Displays a menu of all properties associated with your report. This menu is populated with the items that you define with the **SmartPlant Reports** toolbar **Define** command. The **Map Properties** menu contains a subset of all the properties in the plant database.

You can select a cell in your report template and then assign a property from the **Map Properties** list. The software places the corresponding property in the selected cell. You do not have to map all the properties in the **Map Properties** menu, and you can add properties to the menu by using the **Define** command again.

Note

- The **SmartPlant Reports** toolbar appears in Microsoft Excel when you create or edit a report template.

Related Topics
- *Create a New Blank Report Template*, page 418
- *Create a Report Template Based on an Existing Template*, page 419
- *Defining the Contents of Your Report: An Overview*, page 426
- *Defining the Layout of Your Report: An Overview*, page 420
- *Edit a Report Template*, page 431
- *Generating Reports: An Overview*, page 386

Windows and Dialog Boxes

Description Dialog Box

This dialog box allows you to enter a long text value in the **Description** text box in the **Common Properties** dialog box for an item. For long descriptions, click the ellipsis button [...] to bring up the **Description** dialog box.

Description - Allows you to enter a description for the item.

Note

- Very long descriptions cannot be displayed in their entirety in labels, reports, and so forth.

Related Topics
- *Review and Edit Item Properties*, page 110

Find Dialog Box

Allows you to search for a specific item based on user-defined search criteria. The items that are available depend on the object from which you invoke the **Find** command.

Tag - Enter the tag name of an item for which you want to search. You can type an asterisk (*****) as a wildcard character to find multiple characters, or type a question mark (**?**) as a wildcard character for a single character.

Equipment type - Specifies the type of item for which you want to search. When you use **Find** from the **Reference Data Explorer**, the equipment type that appears is **Ref Plant Item**; when you use **Find** from the **Electrical Index** or the **Electrical Engineer**, the equipment type that appears is **Plant Item**. In specific dialog boxes, where you use the **Find** dialog box to associate items, the software displays only the equipment type appropriate for the item that you select.

Layout - If one or more layouts are defined in the Tabular Editor for the selected equipment type, you can select a specific layout to specify the property columns that appear in the list of results. If you do not specify a layout, the software displays the results using the default columns.

Properties

Show only properties that have data values - Select to filter the list of properties so that only those properties that have data values are displayed.

✎ **Note**

- You can display properties alphabetically or by categories by clicking the corresponding display mode button. Click ᴬ↓ to display alphabetically. Click ▤ to display by categories.

Property - Displays the properties of the selected equipment type.

Value - Enter the value of the property that appears in the **Property** box to search for items that have that value only. You can type an asterisk (*****) as a wildcard character to find multiple characters, or type a question mark (**?**) as a wildcard character for a single character. If the property has values that appear in a list, select the desired value from the list.

Results

Displays the search results with a list of the items that correspond to the criteria that you entered in the upper part of this dialog box.

No. - The sequence of the item as it appears in the list of results.

Item Tag - The tag name of the item.

Equipment Type - The equipment type to which the item belongs.

Find Now - Click to start a search of the items that correspond to the equipment type, and where relevant, the other filter criteria that you select for the search.

New Search - Click to clear all search criteria in preparation for a new search.

Results - Displays a list of the items that meet the search criteria. Highlight an item on the list and select it by doing one of the following:

- Press the **Enter** key.
- Double-click the item.
- Where you are allowed to select more than one item, for example when adding typical control stations to a motor profile, press the **Ctrl** or **Shift** keys to make multiple selections, and then click **OK** or **Apply**.

OK - Click to navigate to or select items that are highlighted in the **Results** window and close the **Find** dialog box.

Cancel - Click to close the **Find** dialog box without navigating to or selecting highlighted items. You can also press the **Escape** key to cancel the operation.

Apply - Click to navigate to or select items that are highlighted in the **Results** window. This option is useful when it is possible to select more than one item and you want to do so without closing the dialog box.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply Lookup Table Data to Equipment*, page 143
- *Associate a Control Station with an Item*, page 226
- *Associate a Control Station with Cables*, page 227
- *Associate a Non-Power Cable with Electrical Equipment*, page 230
- *Associate External Documents with Electrical Items*, page 381
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Create a Control Station*, page 176
- *Create a Profile*, page 61
- *Find Other Parent Items in the Electrical Engineer*, page 91

Move Items to Plant Group Dialog Box

This dialog box appears when you click **Actions** > **Move Items** and it allows you to move one or more selected items to a specific plant group within a plant.

Expand the hierarchy to the desired plant group type and select the plant group to which you want to move the items.

New Item Dialog Box

This dialog box appears when you select an item type in the **Reference Data Explorer**, in the **Electrical Index**, or in the Tabular Editor, and you click **File** > **New** > **Item** where it is possible to create more than one type of new item. The dialog box displays a list of available item types from which you select the desired item type for the new item.

Available items - Displays a list of item types. The available item types depend on the selection you made prior to opening the dialog box. Select the desired item type from the list.

Related Topics
- *Create a Project Item*, page 53
- *Item Tag Dialog Box*, page 720

Open Plant Group Dialog Box

This dialog box appears when you click **File** > **Open** > **Plant Group** and it allows you to navigate to a specific plant group within a plant.

Expand the hierarchy to the desired plant group type and select the plant group where you want to work.

Select Plant - Opens a dialog box with a list of plants for selection.

Related Topics
- *Connect to a Plant Group*, page 33
- *Connecting to a Plant Group: An Overview*, page 32

Open Plant Structure Dialog Box

Displays the structure of the plant to which you are connected. Enables you to set options for connecting to a site with a different plant structure.

Available plant structures - Lists the available plant structures. You can select only one plant structure at a time.

Application type - Displays the application associated with this database.

Open - Connects you to the selected database and checks whether you have the correct access privileges for the selected plant.

Site Server - Opens the **Open Site Server** dialog box, where you can select a SmartPlant initialization file from local and network directories. Plants that correspond to the initialization file you choose are subsequently displayed in the list of available plants in the database.

Preferences Dialog Box

Allows you to customize default preferences for various general options, preferences related to single line diagram (SLD) options, PDB load summary generation settings, and delete loads settings.

Related Topics
- *Customizing Preferences: An Overview*, page 113
- *Preferences Common Tasks*, page 114

General Tab (Preferences Dialog Box)

Allows you to customize default preferences for creating and duplicating new items, displaying PDB item tags in the **Electrical Engineer**, and activating log file generation.

Open after creating or duplicating an item

Allow you to customize a preference for software response after creating or duplicating an item.

None - Displays a created or duplicated item in the **Electrical Index** or the **Reference Data Explorer** without opening either the **Item Tag** or **Common Properties** dialog box, with a default property value and naming convention defined in Options Manager.

Item Tag dialog box - Opens the **Item Tag** dialog box that allows you to type the item tag name for the new item. The text can either be free text or a mask that fits the item naming convention.

Common Properties dialog box - Opens the **Common Properties** dialog box for the created or duplicated item, where you can enter electrical properties as you require.

Electrical Engineer display options for buses, incomers, and bus risers

Allow you to display buses, incomers, and bus risers in the **Electrical Engineer** showing the names of the associated PDB item tags.

Include associated PDB item tags - Displays in the **Electrical Engineer** buses, incomers, and bus risers together with the associated PDB item tags. For example, MCC 200 \ Bus A.

Log file options

Allow you to generate various error log files to display errors that can occur during a batch operation, such as batch generation of schematics, batch cable sizing, batch deletion of items, batch load assignment, and so forth.

Activate log file generation - Enable log file generation options.

Related Topics
- *Preferences Common Tasks*, page 114
- *Preferences Dialog Box*, page 583
- *SLD Tab (Preferences Dialog Box)*, page 584

SLD Tab (Preferences Dialog Box)

Allows you to specify various preferences for single line diagrams (SLD).

SLD drawings

Allow you to set general preferences for SLD generation.

Save settings of last generated drawing - Saves and then applies all the settings defined on the **Single Line Diagram Options** dialog box apart from the settings defined on the **Preferences** dialog box (**SLD margins** and **Distance for SLD internals**). This preference option takes effect after generating an SLD and applies to all subsequent SLD generations. Clearing the **Save settings of the last generated drawing** check box saves only the SLD margin preferences and returns all the other SLD options to the application defaults.

Use bus tie symbols - This option deals with the way the software represents the connection between two redundant buses in a power distribution board.

- Select this option to show a horizontal representation between two buses using the BusTie.sym file. In this case, the software shows the connection between the two buses as a straight bus-tie connection without showing the items connected between and within the circuits. Note that the BusTie symbol is a "circuit" type symbol and has its first connection point on the rightmost side and the second connection point at the leftmost side. Also, the item property in the SLD displays only the coupler information. Furthermore, if you use the **Find in SLD** command, the software finds the pertinent coupler and not the bus riser.

- Clear this option to use a vertical representation of a connection between two buses. This is the standard bus coupler-riser circuit connection representation that includes both coupler and the riser circuits and displays them as vertical objects, connected to each other. The software displays both circuits as standard arrows.

Retrieve last saved position of electrical items - This option deals with the way the software shows the position of electrical items in an SLD that you open from the **Documents** folder in the **Electrical Index**. Every time you open an existing SLD, the software regenerates the SLD by retrieving the relevant data from the database. The software optimizes the positioning of the SLD elements in the drawing every time the SLD is generated. If you move an electrical item and then save the SLD, the software can open the saved SLD showing the last saved position of the electrical items or regenerate the SLD and thus optimize the positioning of the electrical items. Note that this option affects the positioning of electrical items only. The software always retains the position of redlining, annotation, and legend elements as they were saved the last time.

- Select this option to show the position of electrical items as they were in the last saved drawing.
- Clear this option to revert to the optimized positioning of electrical items by regenerating the SLD.

SLD margins

Define the default unit of measure for SLD margins and distance between the page edges and the SLD drawing so that the drawing does not overlap the title block and the SLD border.

Units - Select the unit of measure for the SLD margins (mm or inches). The unit of measure for top, bottom, left and right margins changes dynamically, according to the selection that you make.

Top - Allows you to type the value for the top drawing margin.

Bottom - Allows you to type the value for the bottom drawing margin.

Right - Allows you to type the value for the right drawing margin.

Left - Allows you to type the value for the left drawing margin.

Distances for SLD internals

Set a distance between buses, circuits and circuit internals in a generated SLD.

Distance between buses - Allows you to type a value for the horizontal distance between the buses and select a unit of measure.

Distance between circuit internals - Allows you to type a value for the vertical distance between the circuit internal components (fuses, starters, and so forth) and select a unit of measure.

Distance between circuits - Allows you to type a value for the horizontal distance between the circuits above and below the buses and select a unit of measure.

Reports Tab (Preferences Dialog Box)

Allows you to specify the PDB loads that will account for the coincidence factors and whether and how to account for any loads connected to redundant buses.

Calculation method for All Feeder Load Summary Report

Allows you to specify the coincidence factor to be used in the calculation of the total connected loads.

Use each load coincidence factor - Uses the coincidence factor of each individual load to calculate the total connected loads on the bus.

Use Bus PDB coincidence factor - Uses the average coincidence factors of the bus to calculate the total connected loads on the bus.

Inclusion of coupled buses in All Feeder Load Summary Report

Allows you to specify inclusion options of coupled buses in the PDB load summary calculation.

Do not include coupled buses - Includes in the calculation only those loads that are connected to the current bus and does not include the loads that are connected on the redundant buses.

Include coupled buses connected by couplers only - Includes in the calculation the loads connected on the current bus as well as loads connected on all the redundant buses that are connected to the current bus by a coupler circuit.

Include coupled buses connected by couplers and risers - Includes the loads of the current bus as well as the loads on all the redundant buses that are connected to the current bus by coupler and riser circuits.

Account for converting equipment power losses - When running bus load calculations, allows you to instruct the software to calculate the total electrical consumption taking into account converting equipment power losses. The software adds these power losses to the upstream feeding bus and accounts for them as continuous loads. The same losses are also accounted for in the All Feeder Load Summary and PDB Load Summary reports.

Related Topics
- *General Tab (Preferences Dialog Box)*, page 584
- *Preferences Common Tasks*, page 114
- *Preferences Dialog Box*, page 583

Delete Loads (Preferences Dialog Box)

Allows you to customize the **Delete Loads** default settings as set in the Options Manager. You use these settings to select which related (sub) items of a main item are deleted when deleting a main item from the Electrical Index.

Delete Options

Feeding Power Cables — Allows you to delete feeding power cables.

Associated Control, Grounded, and Instrument Cables — Allows you to delete associated control, ground, and instrument cables.

Associated Control Station/Instruments — Allows you to delete the associated control station and instruments.

Feeding Circuit — Allows you to delete the feeding circuit.

Schematic Document — Allows you to delete the schematic document.

Update Select Lists Dialog Box

This dialog box allows you to add or edit items on SmartPlant Electrical select lists.

Sort - Opens the **Sort** dialog box. You can sort select entries based on entries in one or more column.

Move Up - Moves the chosen select entry row up one position.

Move Down - Moves the chosen select entry row down one position.

Selected list - Allows you to select a select list for updating. You can define new select lists in the Data Dictionary Manager.

Dependent list - Indicates that the value in the selected list depends on a particular selected value in another list. For example, if you choose **Model** from the selected list, the dependent list value is **Manufacturer**. If a list has no dependent list related to it, the value **None** appears.

Disable - Select to hide the select list entry in the user interface. Clear to make the entry available to users.

Value - The name of the select list entry in the column. You can add select list entries by typing in the **Value** column of the last row of the selected list. All entries in a particular select list must have unique values, and you cannot delete entries in select lists.

Short Value - A shortened name for the select list entry, if necessary, in the column. The short value is the one that appears in drop-down lists in the user interface.

Dependent Value - Provides a way to link related select lists, if a dependent list is available. For example, **Electrical Equipment Type** is dependent on **Electrical Equipment Subclass**, which is in turn dependent on **Electrical Equipment Class**. So the range of values that you see in the **Properties** window is limited to values defined for a dependent list, if one is specified for the given property. For instance, motors belong to the **Electrical Equipment Subclass: Motor** and to **Electrical Equipment Class: Electrical Equipment**. The available values in the **Electrical Equipment Type** select list, such as **Squirrel Cage**, apply only to motors, and do not appear in lists for other electrical equipment subclasses.

Sort Dialog Box

Allows you to sort select list entries by multiple combinations of **Value**, **Short Value**, and **Dependent Value**, and then by ascending or descending order within those criteria. This dialog box opens when you click **Sort** on the toolbar of the **Update Select Lists** dialog box.

Sort Items by - Specifies the value and the order that you want to display items in the drawing software.

Ascending - Sorts the selected items in order from the beginning of the alphabet or the lowest number, depending on the type of entries in the select list.

Descending - Sorts items starting from the end of the alphabet or the highest number, depending on the type of entries in the select list.

Then by - Sorts items by an additional value. For example, if you want to sort a select list by its dependent values, you could sort entries with the same dependent value by their **Short Value** or **Value**.

Clear All - Resets all the sorting criteria to none, removing any changes that you made.

Associated Drawings Dialog Box

Allows you to view all drawings that are associated with a specific item in the **Electrical Index,** filter them by drawing type, and then select and open a drawing.

Filter by drawing type

Filters the drawings by their types. The five types are:

- All drawings
- Schematics
- Wiring diagrams
- Cable block diagrams (CBD)
- Single line diagrams (SLD)

Associated Drawings

Document — Displays the name of the document.

Description — Displays a description of the document. (Only available if a description was entered in the common properties of the selected item).

Drawing Type — Displays the drawing type.

Open Drawing — Opens the selected drawing.

Apply Reference Data to Project Cables Dialog Box

Enables you to update data in project cables in batch mode where the reference data for those cables has changed.

Data window - Displays project cables for which data has changed in the reference cables associated with those project cables, with details of the cable category, cable tag, cable formation, and the reference cable. Beside each cable name is a check box that allows you to select the cable for applying the new reference data. For a parallel cable assembly, the software displays only the first cable of the assembly; however it applies the reference data to all the cables in the assembly.

Select all - Allows you to select all the cables in the list for applying updated reference data.

Apply - Applies the new reference data for the selected cables.

Close - Closes the dialog box.

Compare - Opens the **Compare Project and Reference Cable Data** dialog box, from which you can compare for the selected project cable the properties that have different values in the project and reference cables.

Drum and Cable Options (Assign Drums to Cables Wizard)

Choose options on this page to specify initially which cables you want to assign to drums and how you want the software to deal with drum creation and deletion. You must specify one or all of the reference cables and one or all of the pulling areas to be able to proceed to the next step.

Include assigned cables - Select this check box if you want to revamp your entire drum - cable assignments. When you select this option, the software dissociates any cables that are already assigned to drums, and then assigns all the cables. Clear the check box to assign only those cables that are not already assigned to drums.

Create new drums - Select this check box to create new drums automatically for each cable type if the drum - cable assignment results in all existing drums being used. The software creates new drums using the defined naming conventions. If you clear this option, the software assigns the cables to existing drums only and leaves the remaining cables unassigned until you create new drums manually.

Delete empty drums - Select this check box so that if reassignment of cables results in empty drums, the software will delete those drums.

Reference cable - Click the ellipsis button [...] to select a reference cable used to specify the types of cables that you want to assign to drums automatically, or select the **Select all reference cables** check box to specify all the cables of all types in the project.

Pulling area - Select a pulling area or select the **Select all pulling areas** check box to assign cables in all the pulling areas.

Related Topics
- *Drum - Cable Assignment Summary (Assign Drums to Cables Wizard)*, page 592
- *Select Cables (Assign Drums to Cables Wizard)*, page 591

Select Cables (Assign Drums to Cables Wizard)

The displayed cables are those that meet the criteria that you selected on the **Drum and Cable Options** wizard page. When you select a group of cables and click **Next**, the software orders the cables by type (specified by the combination of cable category, cable specification, and cable formation) and by pulling area. Then for each cable type per pulling area, beginning with the longest cable, the software assigns the cables to the drums, beginning with the drum that has the highest available capacity. The software continues to assign cables to the drum provided the total cable length on the drum does not exceed the maximum drum capacity. If assigning a particular cable to a drum would result a total cable length that is greater than the maximum drum capacity, the software assigns that cable to the next available drum.

Notes

- To view this page in full-screen mode so that you can see more of the cable properties, click [icon].

- If you need to add or modify data values for a particular cable, you can do so by selecting the desired row and then clicking **Common Properties**. You can then edit the cable data as needed.

- You can define and customize pulling areas in the Data Dictionary Manager.

Select - Select the check box beside each cable that you want to include in the automatic drum assignment. You can only select a cable for assignment to a drum if a cable length and pulling area are defined.

Cable Tag - The identifying tag of the project cable.

Cable Category - Displays the category that defines the purpose for which the cable is used: **Power**, **Instrumentation**, **Control**, or **Grounding**.

Cable Specification - Displays the family of reference cables to which the reference cable associated with the project cable belongs.

Cable Formation - Displays the cable formation (number and size of conductors) for the reference cable.

Actual Length - Displays the actual length of the cable, as calculated by the software.

Pulling Area - Displays the pulling area assigned to the cable.

Select all - Selects this check box to select all the available cables for inclusion in automatic drum assignment, provided that the cable length and pulling area are defined. Clear to clear the selection for all the cables.

Common Properties - Opens the **Cable Common Properties** dialog box for the selected cable to allow you to update the cable properties.

Related Topics
- *Drum - Cable Assignment Summary (Assign Drums to Cables Wizard)*, page 592
- *Drum and Cable Options (Assign Drums to Cables Wizard)*, page 591

Drum - Cable Assignment Summary (Assign Drums to Cables Wizard)

This page provides you with a summary of the drum - cable assignment for the cables you selected on the **Select Cables** wizard page. The summary includes the actual length of cable assigned to each drum, the cable drum to which each cable was assigned, and the total length of cable assigned to the drum. If the software created a new drum, the row appears in bold type. Bold gray text indicates drum - cable assignments that failed, and the reason for the failure appears in the **Comments** column.

Note

- To view this page in full-screen mode so that you can see more of the cable properties, click.

Related Topics
- *Drum and Cable Options (Assign Drums to Cables Wizard)*, page 591
- *Select Cables (Assign Drums to Cables Wizard)*, page 591

Batch Cable Side and Gland Associations Dialog Box

Allows you to associate one or both sides of a group of non-power cables with a specific item such as a cabinet. You can also assign glands of the same type to a group of cables belonging to any category.

From

Describes information relating to the connection point for one side of the cable.

Tag - Indicates the name of the item associated with this cable side. If you associate the **From** cable side with a circuit that belongs to a PDB, the software displays the path of the circuit tag: PDB/ bus/(cell)/ circuit.

Description - The description is taken from the description of the tag associated with the **From** side of the cable and is read-only.

Dissociate existing connection - Click to dissociate an item that already has an association with this cable side.

Find - Click to open the **Find** dialog box, where you can select an appropriate item to associate.

Clear - Click to clear the selection that appears in the **Tag** and **Description** boxes.

Gland - Displays a gland associated with this cable side, if appropriate.

Dissociate gland - Click to dissociate a gland that already has an association with this cable side.

Find - Click to open the **Find** dialog box, where you can select an appropriate cable gland to associate.

Clear - Click to clear the selection that appears in the **Gland** box.

To

Describes information relating to the connection point for the opposite side of the cable to the **From** side.

Tag - Indicates the name of the item associated with this cable side. If the item that you associate is connected to a PDB/Cell/Circuit, the software displays the association on the **From** side.

Description - The description is taken from the description of the tag associated with the **To** side of the cable and is read-only.

Dissociate existing connection - Click to dissociate an item that already has an association with this cable side.

Find - Click to open the **Find** dialog box, where you can select an appropriate item to associate.

Clear - Click to clear the selection that appears in the **Tag** and **Description** boxes.

Gland - Displays a gland associated with this cable side, if appropriate.

Dissociate gland - Click to dissociate a gland that already has an association with this cable side.

Find - Click to open the **Find** dialog box, where you can select an appropriate cable gland to associate.

Clear - Click to clear the selection that appears in the **Gland** box.

Cable Tag - Displays the name of the cable. Select the check box beside the cable tag to select the cable for associating with specified items and glands.

Description - Displays the description for the cable, if defined.

Cable Category - Displays the cable category: **Control**, **Instrumentation**, or **Grounding**.

Cable Specification - Displays the cable specification, if defined.

From - Displays the item currently associated on the **From** cable side, if defined.

To - Displays the item currently associated on the **To** cable side, if defined.

Related Topics
- *Associate Cable Sides with Glands*, page 193
- *Batch Associate Cable Sides with Glands*, page 194
- *Batch Dissociate Cable Sides and Glands*, page 194
- *Batch Replace Cable Side Associations*, page 192
- *Managing Cables Common Tasks*, page 181

Batch Size Cables Window

Enables you to size cables in batch mode.

Data window - Displays the cables that you selected in the **Electrical Index**, with details of the cable tag, cable formation, input sizing data, and where applicable, sizing results. Beside each cable name is a **Select** check box that allows you to select the cable for sizing.

Select all - Allows you to select all the cables in the list for sizing.

OK - Click to perform the sizing for the selected cables and close the dialog box.

Apply Sizing - Click to perform the sizing for the selected cables.

Excel Report - Click to display the data in this dialog box in a Microsoft Excel file. This is useful if you want to format the layout or print out the data.

Related Topics
- *Batch Size Cables*, page 186
- *Managing Cables Common Tasks*, page 181

Cable Common Properties Dialog Box

Enables you to set options for defining general cable properties, design details, connections, earth loop impedance, and cable sizing data.

Related Topics
- *Create a Project Cable*, page 182
- *Managing Cables Common Tasks*, page 181
- *Size a Cable*, page 184

General Tab (Cable Common Properties Dialog Box)

Sets options for general properties that identify and characterize the cable. An asterisk beside a property name indicates required data.

✪ Important

- In your project, you must create new cables based on existing reference cables.

Cable tag - Identifies the cable. The cable tag appears beside the cable in the **Electrical Index**. You must enter a value that matches the *naming convention* for the cable. You define naming conventions in the Options Manager.

Description - Allows you to type a description of the cable, for information only. Note that for a parallel cable assembly, you can edit the description of each cable independently.

Cable category - Defines the purpose for which the cable is used. The cable category can be power, control, grounding, or instrumentation. The software automatically assigns the cable category according to the folder in which you create the cable, and you cannot change the value.

Cable specification - Indicates the family of cables to which the reference cable belongs. You must select a value for this property. When sizing a cable, the software searches only for those reference cables that belong to the selected cable specification. You can define and customize cable specifications in the Data Dictionary Manager.

Reference cable - Defines the reference cable from which you create the new project cable in the **Electrical Index**. The available reference items depend on the cable specification and whether the **Availability** cable property is set to **True**. When you select a reference item, the software completes all the properties of the cable automatically, and you cannot edit the values. For a new power cable, the default value is **Non-sized power cable**, meaning that you can enter values for the sizing data. You must select a value for the **Reference cable** property.

Cable usage - If required, select a value for this property to further categorize the cable, for example, for control voltage definitions, level definitions, and so forth. You can define and customize cable usages in the Data Dictionary Manager.

Armor type - Specifies the type of armor used to protect the cable from its environment. For example, types of armor include non-armored, steel, or lead. This read-only value comes from the reference cable used as the basis for this cable.

Outer jacket insulation - Specifies the material used for the cable jacket. For example, insulation materials include polyvinyl chloride (PVC) and polyethylene. This read-only value comes from the reference cable used as the basis for this cable.

Notes - Allows you to type user notes, for example if you want to provide additional information regarding special features of the cable.

Construction

Describes the physical characteristics related to the construction of the cable. All the properties in this section except for the **Conductor arrangement** and **Color pattern** properties are read-only. The available values depend on the conductor arrangement that you select.

Conductor arrangement - Defines the number and arrangement of the conductors in the cable (not applicable for instrumentation cables). Your selection filters the list of available reference cables to display only those cables that match the cable formation that you have selected. When sizing a power cable, the software searches only for those reference cables that have the selected conductor arrangement. The following standard options are available:

- **Single core cable** - Creates a current-carrying cable with a single core. For this option, you can specify an arrangement of any number of single-core conductors, for example, 3 single-core cables, but you cannot specify any additional conductors.

- **2-core cable** - Creates a cable with two current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **2+1-core cable** - Creates a cable with two current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3-core cable** - Creates a cable with three current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **3+1-core cable** - Creates a cable with three current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3+2-core cable** - Creates a cable with three current-carrying conductors and two additional conductors. You cannot modify the cable construction for this option.

- **Multicore cable** - Creates a cable with multiple conductors. For this option, you can specify any number of conductors and additional conductors.

Note

- You can change the names of the standard conductor arrangement items in the Data Dictionary Manager. However, the software performs calculations according to the position in the list of the item

that you select. For example, a 2-core cable is always the second item in the list.

Set type - For instrumentation cables only, allows you to select a standard value that represents the arrangement of the wires in the set, such as **Wire**, **Pair**, **Pair w/Shield**, **Triad**, **Triad w/Shield**, and so forth. You can select values of this property for a new cable only; you cannot select a different value for an existing or duplicated cable. If the cable sets are not all of the same type, the value **Other** appears.

Cable formation - Displays computed information based on the total number and the size of the conductors or cable sets specified. The size appears as a question mark when the reference cable type is **Non-sized power cable** or when you have not sized a power cable.

Current-carrying conductors - Specifies the number of conductors within the cable used for carrying the current, excluding the neutral, the ground, or additional conductors (not applicable for instrumentation cables). If you selected **Single-core** or **Multicore** as the conductor arrangement, this option allows you to specify the number of conductors that carry the electric current.

Number of sets - For instrumentation cables only, allows you to type the number of cable sets that the cable contains. Click the ellipsis button ⬚ to open the **Cable Set Properties** dialog box where you can modify the set definitions if required.

Size - Displays the standard size for the current-carrying conductors or cable sets.

Material - Specifies the conductor material used for the current-carrying conductors or cable sets. This read-only value comes from the reference cable used as the basis for this cable. For power cables, the software uses this property in conjunction with the **Conductor Material Coefficients** lookup table for performing short circuit cable sizing.

Insulation - Specifies the material used for insulating the current-carrying conductors or cable sets. For example, insulation materials include polyvinyl chloride (PVC) and polyethylene. This read-only value comes from the reference cable used as the basis for this cable. For power cables, the software uses this property in conjunction with the **Conductor Insulation Temperatures** lookup table for performing short circuit cable sizing.

Additional conductors - Depending on the selected conductor arrangement, this property specifies the number of additional non-current-carrying conductors, such as ground wires (not applicable for instrumentation cables).

Overall shield - For instrumentation cables only, select to specify an overall shield for the cable. When you select this option, you need to specify the size and material of the overall shield.

Size - Displays the standard size for the additional conductors or overall shield.

Material - For instrumentation cables only, allows you to select the material used for an overall shield. You can define and customize the available materials in the Data Dictionary Manager.

Communication wire - For instrumentation cables only, select to specify a communication wire for the cable.

Color pattern - Select a color pattern for the cable from one of the patterns available. Click the ellipsis button to define or modify a color pattern.

Sizing required - For power cables only, select this option to enable the **Sizing Data** tab so that you can size the cable. When you create a new power cable, the software selects this check box by default.

Use for neutral grounding - For grounding cables only, select this option to make the cable available for selection as a grounding cable on the **Transformer Connections and Tapping** dialog box.

Insulation voltage rating - Shows the voltage that corresponds to the manufacturer voltage rating for the cable. This read-only value comes from the reference cable used as the basis for this cable.

Insulation temperature rating - Shows the temperature that corresponds to the manufacturer rating of the maximum operating temperature for the cable. This read-only value comes from the reference cable used as the basis for this cable.

Basic ampacity in air - Shows the ampere capacity of the cable in air under the basic conditions that the manufacturer specifies. The software uses this property in the sizing algorithm when you select **In Air** as the value for the **Cable routing** property on the **Sizing Data** tab. This read-only value comes from the reference cable used as the basis for this cable.

Basic ampacity in ground - Shows the ampere capacity of the cable in the ground under the basic conditions that the manufacturer specifies. The software uses this property in the sizing algorithm when you select **Under Ground** as the value for the **Cable routing** property on the **Sizing Data** tab. This read-only value comes from the reference cable used as the basis for this cable.

Reactance per unit length - Shows the value of reactance per unit length that matches the manufacturer specification. Reactance per unit length applies only to cables that carry alternating current (AC). The software uses this property in the sizing algorithm. This read-only value comes from the reference cable used as the basis for this cable.

Resistance per unit length - Shows the value of resistance per unit length that matches the manufacturer specification. The software uses this property in the sizing algorithm. This read-only value comes from the reference cable used as the basis for this cable.

Parallel cables

Describes the number of cables in parallel and the list of cable names. The software calculates these values based on the result of a cable sizing operation or you can specify the number of parallel cables manually in order to create a parallel cable assembly.

Total number - Specifies the number of parallel cables where the current cable is part of a cable assembly that shares the load current and constitutes an inseparable group. When you perform and apply cable sizing where the recommended number of parallel cables shown on the **Sizing Data** tab exceeds the value in this box, the software copies the calculated value to this box after you click **Apply**. You can also select the number of parallel cables manually without performing sizing. To specify a single cable, select the blank value. In the Options Manager, you can specify the minimum conductor size and the maximum number of parallel cables allowed.

List - When generating parallel cables, either through cable sizing or when you increase the total number of parallel cables, the software creates the additional cables based on the original cable name and adds a suffix. In the Options Manager, you can specify whether to use a numeric or alphanumeric suffix by default; you can overwrite the default suffix for each parallel cable by typing a value for the **Parallel Cable Suffix** property in the **Properties** window. Note that if you rename a parallel cable in an assembly, the software does not update the **Parallel Cable Suffix** property for that cable. These cables behave as individual cables, so that if you delete one of the cables, the software leaves the other cables intact. However, if you change a parameter for one of the cables, the software updates the same parameter for all the cables. You can select any cable from the list to view its parameters, and the name of the cable in the **Cable tag** box changes accordingly.

Compare to Reference - Click to open the **Compare Project and Reference Cable Data** dialog box, from which you can compare reference cable properties that have changed since the creation of the project cable. From this dialog box, you can also copy the updated reference data to the project cable if required.

Related Topics
- *Create a Project Cable*, page 182
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Design Data Tab (Cable Common Properties Dialog Box)

Provides information about the cable drum assignment, routing, length, and so forth. An asterisk beside a property name indicates required data.

Drum assignment

Allows you to assign the current cable to a cable drum or to unassign the cable from a drum.

Drum - Displays the cable drum to which you assigned the current cable. Note that this box is available only if the length of the current cable does not exceed the maximum cable drum capacity.

Number of splices - Displays the number of splices that the software created when you assigned the current cable to a drum. Note that this box is available only if the length of the current cable exceeds the maximum cable drum capacity.

View - When the software shows the number of splices, allows you to open the **Cable Splices** dialog box where you can see the cable cuts and their lengths according the assigned drums.

Find - Opens the **Find** dialog box, where you can select a cable drum to which you assign the current cable.

Unassign - Cancels the assignment of the current cable to the cable drum displayed under **Drum**.

Pulling area - Select a pulling area to use when routing the cable from the selected drum. You can define and customize pulling areas in the Data Dictionary Manager.

Maximum drum capacity - This information only applies to reference cables.

Minimum cable length to order - This information only applies to reference cables.

Cable routing

In this group box, you build the routing path for the cable that you are editing.

Select predefined route - When you click **Find**, allows you to select a predefined route to use as a basis for the cable routing definition.

Copy route from cable - When you click **Find**, allows you to select routing from another cable to use as a basis for the cable routing definition.

Segregation level - You must select a *segregation level* for the cable if you want to perform routing. The segregation level of the cable must match the segregation level of all the cableway segments to which you assign the cable. Note that once you have added at least one segment to the routing, you cannot change the segregation level; to do so, you must first remove all the segments.

Find - opens the **Find** dialog box, from which you can select as predefined route or a cable with routing for copying, according to the option you have selected. Note that only routes or cables that belong to the current segregation level are available for selection.

Save as new route - Allows you to save the current routing definition as a new predefined route.

Route details

Displays the cableways and segments that make up the routing. Note that only those segments that belong to the current segregation level are available for selection.

Sequence - Indicates the sequence of the segment in the routing path.

Cableway - Select the cableway from which you want to select a segment.

Segment - Select the segment through which you want to route the cables. You can use each segment only once in a route.

Length - Displays the segment length for the selected segment. You define this value in the **Properties** window.

Add - Adds an empty row below the currently selected row.

Insert - Adds an empty row above the currently selected row.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order of the segments in the route.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order of the segments in the route.

Remove - Removes the currently selected segment from the route.

Design data

Provides information about the physical characteristics of the cable.

Use external routing length - Select to use the value of the **External routing length** property, if a value exists. If you select this check box, and the **External routing length** property does not have a value, the software uses a value of zero in calculations. If you clear this check box, the software uses the value of one of the following: **Estimated length**, **Design length**, or **Routing length**.

Estimated length - Allows you to enter the estimated length of cable before installation with appropriate units of measure. In sizing calculations, the software uses this value if you do not specify a design length. You can specify the default project units of measure in the Data Dictionary Manager.

Design length - Allows you to enter a more accurate length for the cable after installation with appropriate units of measure. In sizing calculations, this value overrides the estimated length. You can specify the default project units of measure in the Data Dictionary Manager.

Routing length - If you added routing information, displays the total length of the segments that you defined.

External routing length - If you retrieved routing information from SmartPlant 3D, the software can use the routing length obtained with the retrieved data, provided that you select the **Use external routing length** check box.

Tail 1 - Type a value for the spare tail length at one end the cable, if you require.

Length - A calculated field that displays the design length, or if not available, the estimated length.

Tail 2 - Type a value for the spare tail length at the other end the cable, if you require.

Spare length - Type a value for additional spare cable length, if you require.

Actual length - The actual length of the cable, calculated from the sum of tail length 1, the main length of the cable (routing length, design length or estimated length), tail length 2, and the spare length.

Note

- For parallel cables, when you change estimated cable length or design cable length, the software automatically changes the values for all of the other parallel cables.

Compare to Reference - Click to open the **Compare Project and Reference Cable Data** dialog box, from which you can compare reference cable properties that have changed since the creation of the project cable. From this dialog box, you can also copy the updated reference data to the project cable if required.

Related Topics

Connections Tab (Cable Common Properties Dialog Box)

From

Describes information relating to the connection point for one side of the cable. For a power cable, this property indicates the cable side that is closer to the power source.

Tag - Indicates the name of the item associated with this cable side. If you associate the **From** cable side with a circuit that belongs to a PDB, the software displays the path of the circuit tag: PDB/ bus/(cell)/ circuit.

Description - The description is taken from the description of the tag associated with the **From** side of the cable and is read-only.

Find - Click to open the **Find** dialog box, where you can select an appropriate item to associate.

Dissociate - Click to dissociate an item that already has an association with this cable side.

✍ Notes

- If you make electrical associations in the **Electrical Engineer** or on the **Feeder Data** tab of the **Electrical Motor Common Properties** dialog box, the software populates the **Tag** property on the **From** cable side with the appropriate data.

- If you associate a power cable with a load that you previously associated with a circuit, the software inserts the cable between the circuit and the load and populates the **Tag** property with the circuit data.

- Once you have made an association for a power cable, all further changes, including dissociation, must be made in the **Electrical Engineer**.

Gland - Displays a gland associated with this cable side, if appropriate.

Find - Click to open the **Find** dialog box, where you can select an appropriate cable gland to associate.

Dissociate - Click to dissociate a gland that already has an association with this cable side.

To

Describes information relating to the connection point for the opposite side of the cable to the **From** side. For a power cable, this property indicates the cable side that is closer to a consumer.

Tag - Indicates the name of the item associated with this cable side. For a power cable, if you associate the **To** cable side with a load, the software uses data from the load to populate some of the properties on the **Sizing Data** tab; if the load is connected to a PDB/Cell/Circuit, the software displays the association on the **From** side.

Description - The description is taken from the description of the tag associated with the **To** side of the cable and is read-only.

Find - Click to open the **Find** dialog box, where you can select an appropriate item to associate.

Dissociate - Click to dissociate an item that already has an association with this cable side.

✎ Notes

- If you associate a power cable with a motor by applying a profile, the software populates the **To** property with the motor tag.

- Once you have made an association for a power cable, all further changes, including dissociation, must be made in the **Electrical Engineer**.

Gland - Displays a gland associated with this cable side, if appropriate.

Find - Click to open the **Find** dialog box, where you can select an appropriate cable gland to associate.

Dissociate - Click to dissociate a gland that already has an association with this cable side.

Related Topics
- *Create a Project Cable*, page 182
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *Earth Loop Impedance Tab (Cable Common Properties Dialog Box)*, page 607
- *Procurement Tab (Cable Common Properties Dialog Box)*, page 612
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Sizing Data Tab (Cable Common Properties Dialog Box)

Provides information about the electrical load for a power cable. The software uses this information as input values for cable sizing. When the cable has a connected load, the software copies the electrical parameters of the load to the cable. You can change the values of these parameters used for cable sizing if required. An asterisk beside a property name indicates required data for sizing.

Supply - Select the type of supply: alternating current (AC) or direct current (DC). If the cable is connected to a load for which you select the **Consumer** check box on the **General** tab of the appropriate load **Common Properties** dialog box, the software copies the value from the load to this box.

📝 **Note**

- The software performs sizing using a different voltage drop formula for AC and DC supplies.

Rated voltage - Select the appropriate operating voltage for the cable. If the cable is connected to a load for which you select the **Consumer** check box, the software uses the value from the load.

Number of phases - Select the number of phases in the power supply to the load: 1, 2, or 3. If the cable is connected to a load for which you select the **Consumer** check box, the software uses the value from the load. This property applies for an AC supply only.

📝 **Note**

- The software performs sizing for 1- and 3-phase supplies only, using a different voltage drop formula for each option.

Full load current (FLA) - Enter the electrical current, in mA, A, or kA, that the cable must supply without causing damage to the insulation. The software uses the value of the full load current to perform sizing involving the thermal calculation. If the cable is connected to a load for which you select the **Consumer** check box, the software copies the value from the load to this box.

Starting current - If a motor is connected to the cable, the software enters the value of the locked rotor current (LRC). For a motor, the starting current value can typically be 400 - 600% larger than the full load current, and is therefore important when evaluating the current requirements used to perform sizing. Note that this option is available for all power cables whether they are connected to motors or not.

Use code factor - You can specify whether to use an FLA multiplier factor for thermal ampacity calculations. The code factor affects the units of various quantities such as the cable size and the sizing calculation. The standard options defined in the Data Dictionary Manager are:

- **Yes** - The software multiplies the full load current by a factor set by the user in the **Code Factor** field. If **yes** is selected and no value entered in the **Code Factor** field then the software uses a factor of 1.

- **No** - The software does not multiply the full load current.

The selection for this option also determines the available values of conductor size, frequency, and rated voltage.

Code Factor — Allows you to type the value for the FLA multiplier. The code factor value is used to multiply the FLA which is used to calculate the required thermal ampacity, if no value is entered then the software uses a factor of 1. This field is only available when **Use Code Factor** is set to **Yes**.

Power factor at full load - Allows you to enter the power factor of the equipment that the cable feeds under normal full load running conditions. The power factor is the ratio of *active power* to *apparent power*, and can have a value between 0 and 1. If the cable is connected to a load for which you select the **Consumer** check box, the software uses the value from the load.

Power factor at starting - Allows you to enter the power factor of the equipment that the cable feeds under starting conditions. If the cable is connected to a motor for which you select the **Consumer** check box, the software uses the value from the motor. This property is not available for loads other than motors.

Routing - Allows you to select whether the cable routing is above or below ground. The option that you select affects whether the software uses the ampacity value in air or in the ground when performing the sizing calculation.

Auto-restart - Select this option to indicate whether a motor connected to the cable restarts automatically or manually after a power outage. This property is not available for loads other than motors.

Derating factors

The basic ampacity of the cable applies to standard conditions that the manufacturer specifies. Under actual operating conditions, such as a higher environmental temperature, or a location where several cables are routed close to one another, the ampacity changes. The derating and utilization factors take into account the operating conditions of the cable.

Derating factor - Allows you to enter a value for the derating factor, which the software uses to calculate the cable derated ampacity under actual operating conditions. The software multiplies the basic ampacity by the derating factor to obtain the derated ampacity. A value of less than 1 means that the derated ampacity is lower than the basic ampacity, and this will result in a requirement for a larger size cable.

Utilization factor - Enter a value between 0 and 1 to represent an additional factor that derates the cable ampacity. If you do not need to use this property, set the value to 1. The software multiplies the basic ampacity by this factor to obtain the derated ampacity.

Ambient temp. factor - Enter a value between 0 and 1 to represent a temperature factor that affects the derated cable ampacity. For the software to apply this factor to the calculation, in the Options Manager, you must first set the value of the **Apply Temperature Correction Factor to Cable Sizing** property to **Yes**; setting the value of the property to **No** is the equivalent of using a value of 1 for the temperature factor.

Voltage drop (%)

The cable needs to supply electrical current at a voltage that is no more than a few percentage points below the rated voltage. The voltage drop is expressed as a percentage of the rated voltage. The software compares the allowable and calculated voltage drop at running and starting, and uses the result to specify the smallest cable that meets the maximum voltage drop requirements.

Allowable at full load - Enter your maximum percentage voltage drop allowed when the connected load is running under normal full load conditions.

Note

- The allowable voltage drop must have a value greater than zero. To ignore the voltage drop in the sizing calculation, type a value of 100.

Calculated - Displays a calculated value of the percentage voltage drop when the load is running under normal operating conditions, using the estimated or design length of the cable.

Allowable at starting - Enter your maximum percentage voltage drop allowed when starting the equipment. This property is not available for loads other than motors.

Note

- The allowable voltage drop must have a value greater than zero. To ignore the voltage drop in the sizing calculation, type a value of 100.

Calculated - Displays a calculated value of the percentage voltage drop when starting the load, using the estimated or design length of the cable. This property applies to motors only.

Short circuit

If you want to include short circuit calculations in the cable sizing, you enter values for the properties in this section. The software uses this data along with the coefficients that are found in the **Conductor Material Coefficients** and **Conductor Insulation Temperatures** lookup tables to calculate the minimum required conductor cross-section area. Suitable values must be available in these tables for the software to be able to perform the calculation.

Use short circuit cable sizing - Select to enable and activate the short circuit properties and algorithm when you want the cable sizing to include short circuit calculations.

Duration - Allows you to type the maximum time for which the cable is expected to withstand a fault short circuit current until a protection device interrupts the current flow in the circuit.

Fault current - Allows you to type the maximum short circuit fault current that can flow through the cable, and for which the cable needs to be sized.

Required area (cmils) - Displays the calculated minimum conductor cross-section area required for the cable to meet the short circuit criteria that you specify.

Protection setting

Allows you to specify cable sizing based on the rated current for a protection device such as a fuse or circuit breaker, rather than using the load current.

Include protection device - Select to specify the rated current of the protection device for use in sizing calculations. Clear to use the load current value.

Protection settings - Allows you to type the rated current of the protection device for use in sizing the cable.

Sizing results

Shows the outcome of the calculations based on the load data that you supply for the cable. All the information is read-only.

📝 Note

- If the cable is connected to a load for which you select the **Consumer** check box, the software uses the electrical values of the load to size the cable.

Recommended cross section - Displays the recommended cross section of the current-carrying conductors for the load requirement.

Derated cable ampacity - Displays the calculated cable ampacity under actual operating conditions. The software calculates the derated ampacity by multiplying the basic ampacity by the derating factor and the utilization factor.

Number of parallel cables - Indicates the recommended number of parallel cables required when no single reference cable meets the minimum cable sizing requirements. The software creates parallel cables as a result of the sizing. The cables are handled as an inseparable group, and any action that you perform on one of the cables causes all the cables in that group to follow that operation.

Maximum cable length - Indicates the maximum allowed length of cable of the recommended cross-section that meets the current sizing requirements.

Calculation information - Displays messages related verification of the sizing calculation such as missing data or indicating a cable-load association.

Note

- The software displays the message **Not found** when no suitable reference cable matches the sizing results under the following conditions: when no reference cable exists corresponding to the cable specification or arrangement, when there are missing values for ampacity, resistance, or reactance, or when the number of parallel cables exceeds the limit specified in Options Manager by the **Maximum Number of Parallel Cables** property.

System notes - Displays messages related to performed actions such as connecting a load, sizing, and so forth.

Refresh Load Data - Updates any information derived from a load connected to the cable. This button is available when the load data changes since the previous time of sizing of the cable or when you perform cable sizing for the first time.

Perform Sizing - Performs the sizing calculation using the input data and recommends the most suitable reference cable based on your selected cable specification and formation.

Apply Sizing - Copies data to the cable from the reference cable the software recommends as a result of the cable sizing.

Related Topics

Earth Loop Impedance Tab (Cable Common Properties Dialog Box)

This tab appears for project power cables only. You can specify parameters for cable grounding using the cable armor or using a separate grounding cable, as well as external impedance, and the software can calculate the total earth loop impedance for the cable.

Earth loop impedance calculation required - Select this option when you want to calculate the earth loop impedance for this particular line. After you click **Calculate**, the software clears this check box.

Cable impedance path

Allows you to specify the grounding path used for the cable using the cable armor or via a separate grounding cable, if available.

Use cable armor - When you select this option, the software uses the cable armor as the return path for grounding with the values that you specify for the armor resistance and reactance to calculate the earth loop impedance.

Armor impedance per unit length - Displays the overall armor impedance per unit length, which the software calculates from the **Armor Reactance** and **Armor Resistivity** properties as specified in the **Properties** window.

Actual cable length - The value of the **Actual length** property that the software calculates on the **Design Data** tab.

Total armor impedance - The total cable armor impedance, calculated as the product of the armor impedance per unit length and the actual cable length.

Use separate grounding cable - When you select this option, the software uses a grounding cable that is associated with the load as the return path for grounding with the values that you specify for the grounding cable resistance and reactance to calculate the earth loop impedance. Note that this option is only available if a grounding cable is already associated with the load on the **To** side of the power cable.

Tag - The item tag of the grounding cable associated with the load on the **To** side of the power cable.

Cable formation - Displays the cable formation of the grounding cable as specified in the cable common properties.

Cable impedance per unit length - Displays the overall grounding impedance per unit length, comprised of the **Reactance per unit length** and **Resistivity per unit length** properties which are specified for the grounding cable.

Actual cable length - The value of the **Actual length** property for the grounding cable.

Total grounding cable impedance - The grounding cable impedance, calculated as the product of the cable impedance per unit length and the actual cable length.

External loop impedance - Type a value for any additional impedance associated with the power source.

Total earth loop impedance - The total impedance, calculated as the sum of the grounding path impedance, external loop impedance, and impedance of the cable carrying the current.

Calculate - Click to update the calculation of the total earth loop impedance.

System notes - Displays messages related to earth loop impedance calculations.

Related Topics
- *Connections Tab (Cable Common Properties Dialog Box)*, page 602
- *Create a Project Cable*, page 182
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *Earth Loop Impedance Calculation*, page 537
- *Procurement Tab (Cable Common Properties Dialog Box)*, page 612
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Procurement Tab (Cable Common Properties Dialog Box)

Enables you to enter information related to ordering a specific cable for your inventory.

Manufacturer - Allows you to select a manufacturer. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a cable model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Stores part number - Allows you to enter a reference number for the cable in your inventory. This read-only value comes from the reference cable used as the basis for this cable.

Related Topics
- *Connections Tab (Cable Common Properties Dialog Box)*, page 602
- *Create a Project Cable*, page 182
- *Design Data Tab (Cable Common Properties Dialog Box)*, page 597
- *Earth Loop Impedance Tab (Cable Common Properties Dialog Box)*, page 607
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Cable Drum Common Properties Dialog Box

Allows you to define and edit basic cable drum properties, and to assign project cables to the current cable drum. An asterisk beside a property name indicates required data.

Drum tag - Identifies the cable drum. The drum tag appears beside the cable drum in the **Electrical Index**.

Description - Allows you to type a description of the cable drum, for information only.

Reference cable - Displays the reference cable that you selected for the current cable drum. You must select a reference cable for the drum. To select a different reference cable, click to open the **Select Reference Cable** dialog box. Note that you can only select another reference cable if the drum has no assigned cables.

Pulling area - Select the cable pulling area to which you assign the drum. Note that you can only select another pulling area if the drum has no assigned cables. You can define and customize pulling areas in the Data Dictionary Manager.

Minimum cable length to order - Allows you to specify the minimum length of cable when ordering from the supplier. You must enter a value for this property. You can only assign cables whose actual length is greater than the value you specify for this property.

Maximum drum capacity - Allows you to specify the maximum total length of cable that the drum can hold, including any spare cable. The software validates the total cable length on the drum against this value. You must enter a value for this property.

Total cable length to order - Allows you to specify the length of cable to order from the supplier for future assignment. This value must lie between the minimum cable length to order and the maximum drum capacity and cannot be less than the total cable length on the drum.

Total actual length assigned - Displays the sum of the *actual length* values of the assigned cables.

Spare percentage on drum - If you want the software to calculate a spare cable length factor, for example, to make allowance for cutting errors, type the value. For example, if you want a drum spare factor of 5%, type **5**. For calculation details, see *Rules Governing Cable Drum Calculations*, page 212.

Total cable length on drum - Displays the total length of the assigned cables plus the length calculated from the spare percentage value. You can assign cables so long as this value does not exceed the value of the **Maximum drum capacity** property.

Assigned cables

Displays the cables assigned to the current cable drum. Note that you can only assign cables after you have selected a reference cable and a pulling area.

Tag name - Identifies the cable. The tag name appears beside the cable in the **Electrical Index**.

Formation - Describes cable structure in terms of the number of conductors, and where appropriate, additional conductors with their sizes. For example, **3 x 2.5 mm2 + 2 x 1.0 mm2** described a cable with three 2.5 mm^2 main conductors and two 2.5 mm^2 additional conductors.

Actual Length - A derived value for cable length. If in the **Cable Common Properties** dialog box a cable has a value for **Estimated length**, but no value for **Design length**, then the estimated length is the value for **Actual Length**. If there is a value for **Design length**, then this is also the value for **Actual Length**.

Add - available only after you enter the required data — Opens the **Find** dialog box, where you search for available cables to assign to the current cable drum. The available cables are those that belong to the same reference cable and pulling area that you specify for the cable drum.

Remove - available only when cables are associated with the cable drum — Removes the selected cable from the current cable drum.

Related Topics

Cable Set Properties Dialog Box

Provides information about cable sets that you define for an instrumentation cable.

Sequence - Indicates the sequence of the set in the cable. Note that you cannot change the values in this column.

Set Type - The set type represents the arrangement of the wires in the set, such as **Wire**, **Pair**, **Pair w/Shield**, **Triad**, **Triad w/Shield**, and so forth. The default value for this property comes from the set type selected for the cable. You can edit the values of this property so long as you have not assigned a reference cable to a project cable. If the set type is unknown, for example if you change the number of sets in the **Cable Common Properties** dialog box, the value **Other** appears.

Item Tag - Displays the item tag for each set. You can change these values if desired.

Related Topics
- *General Tab (Cable Common Properties Dialog Box)*, page 596

Cable Sizing Matrix Dialog Box

Enables you to review all the acceptable reference cables and parallel combinations defined in the Reference Data Explorer that can be used for the specified load and electrical parameters, and to select from the list a single cable or an assembly of parallel cables. Only those cables that meet or exceed all the sizing criteria appear in the list.

Comment - Indicates whether the suggested cable in the list exactly meets the sizing data criteria or exceeds the needed size as shown by the values: **Recommended** or **Oversized**.

Conductor Size - Shows the conductor cross-section area for each cable that meets the sizing criteria.

Number of Cables - Indicates the number of parallel cables for all appropriate recommended or oversized cables that meet the sizing criteria, up to a maximum determined by the values of the minimum cross-section area and maximum number of parallel cables as specified in the Options Manager.

Maximum Cable Length - Indicates the maximum allowed length of cable of the recommended cross-section that meets the voltage drop requirements.

Required Area (cmils) - This column appears only if you select the **Use short circuit cable sizing** check box and enter short circuit sizing data. It indicates the minimum required conductor cross-section area in cmils needed to meet the short circuit criteria that you specify.

Related Topics
- *Batch Size Cables*, page 186
- *Size a Cable*, page 184
- *Sizing Data Tab (Cable Common Properties Dialog Box)*, page 605

Cable Splices Dialog Box

Displays the cable cuts and their lengths according the assigned drums. The number of splices is always one less than the number of cable cuts.

Sequence - Displays the cable cut sequence.

Drum Tag - Displays the name of the associated drum for each cable cut.

Segment Length - Displays the length of the cable cuts for each drum.

Compare Project and Reference Cable Data Dialog Box

Enables you to compare project and reference cable data where the reference cable data has been changed.

Property - Displays the cable property that has changed. The software compares changes in the values of any of the following properties:

- BasicAmpacityInGround
- BasicAmpacityInAir
- BasicInsulationTemperatureRating
- InsulationThickness
- InsulationColor
- InsulationMaterial
- InsulationTradeName
- InsulationType
- ArmorReactance
- ArmorResistivity
- BendingRadius
- InsulationMaterial

- OutsideDiameter
- InnerSheathDiameter
- InsulationTradeName
- InsulationVoltageRating
- InsulationTempRating
- InsulationThickness
- Resistivity
- Reactance
- InnerInsulationTempRating
- ArmorType
- JacketColor
- JacketInsulationMaterial
- JacketInsulationTempRating
- StorePartNumber
- TemperatureClass
- WeightPerLength
- IngressionProtection
- PullTension

Project Cable Value - Displays the current value of the property in the selected project cable.

Reference Cable Value - Displays the new value of the property in the associated reference cable.

OK - Selects the project cable for copying data in the **Apply Reference Data to Project Cables** dialog box.

Cancel - Closes the dialog box without selecting the cable.

Related Topics
- *Apply Reference Data to Project Cables Dialog Box*, page 590

Compare Project and Reference Cable Data Dialog Box

Enables you to compare project and reference cable data where the reference cable data has been changed. You can copy the updated reference data to the project cable if you require.

Property - Displays the cable property that has changed. The software compares changes in the values of any of the following properties:

- BasicAmpacityInGround
- BasicAmpacityInAir
- BasicInsulationTemperatureRating
- InsulationThickness
- InsulationColor
- InsulationMaterial
- InsulationTradeName
- InsulationType
- ArmorReactance
- ArmorResistivity
- BendingRadius
- InsulationMaterial
- OutsideDiameter
- InnerSheathDiameter
- InsulationTradeName
- InsulationVoltageRating
- InsulationTempRating
- InsulationThickness
- Resistivity
- Reactance
- InnerInsulationTempRating
- ArmorType
- JacketColor
- JacketInsulationMaterial
- JacketInsulationTempRating

- StorePartNumber

- TemperatureClass

- WeightPerLength

- IngressionProtection

- PullTension

Project Cable Value - Displays the current value of the property in the selected project cable.

Reference Cable Value - Displays the new value of the property in the associated reference cable.

Copy Data - Copies the changed data from the reference cable to the project cable. For a parallel cable assembly, the software applies the reference data to all the cables in the assembly.

Cancel - Closes the dialog box without copying the data.

Related Topics
- *General Tab (Cable Common Properties Dialog Box)*, page 596

Define Color Pattern Dialog Box

Enables you to define the colors of each conductor in a cable, and whether the cable consists of a single set of conductors or repeating sets where the cables are identified by the same colors.

Color pattern tag - Select an existing color pattern tag or type a new tag name after clicking **New**.

New - Click to define a new color pattern.

Delete - Click to delete the color pattern that appears in the **Color pattern tag** box.

Pattern type - Click the desired option:

- **Individual colors** - Assign a single color to each conductor. If the number of conductors in the **Cable Common Properties** dialog box is greater than the number of colors you define in this dialog box, the software does not assign colors to the extra conductors.

- **Repeating colors** - Assign color to the conductors according to the sequence of each conductor. If the number of conductors in the **Cable Common Properties** dialog box is greater than the number of colors that you define in this dialog box, the software assigns the colors repeatedly in sequence to all the conductors. This way, you can define colors to apply to multiple sets of conductors.

Wire definitions

Sequence - Indicates the sequence of the conductor in the cable or set.

Color - Select a color for each conductor.

Sequence - Indicates the sequence of the conductor in the cable or set.

Color - Select a color for each conductor.

Add - Adds an empty row below the currently selected row.

Insert - Adds an empty row above the currently selected row.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order of the conductors in the cable or set.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order of the conductors in the cable or set.

Remove - Removes the currently selected conductor.

Related Topics
- *General Tab (Cable Common Properties Dialog Box)*, page 596

Reference Gland Common Properties Dialog Box

Enables you to set options for defining cable glands. When you define project cables, you can add glands to one or both cable ends. The software is able to calculate a gland take-off and generate the results in a report.

General Tab (Reference Gland Common Properties Dialog Box)

Sets options for general properties that identify and characterize the gland. An asterisk beside a property name indicates required data.

Gland tag - Identifies the gland. The reference gland tag is the name that appears beside the gland in the **Reference Data Explorer**.

Description - Allows you to type a description of the gland, for information only.

Gland type - Select a gland type from the list. You can define and customize gland types in the Data Dictionary Manager.

Gland size - Select a standard code used to represent the gland size.

Gland material - Select the material used for the body of the gland.

Plating - Select a material used for gland plating, if appropriate. You can define and customize plating materials in the Data Dictionary Manager.

Explosion protection - Select the type of explosion protection required for the gland, if appropriate.

Ingression protection - Select the type of ingression protection required for the gland, if appropriate.

Design standard - Select a design standard from the list. You can define and customize design standards in the Data Dictionary Manager.

Certification - Select a certification standard from the list. You can define and customize certification standards in the Data Dictionary Manager.

Related Topics
- *Details Tab (Reference Gland Common Properties Dialog Box)*, page 621
- *Procurement Tab (Reference Gland Common Properties Dialog Box)*, page 622
- *Reference Gland Common Properties Dialog Box*, page 620

Details Tab (Reference Gland Common Properties Dialog Box)

Provides information about the structure and dimensions of the gland, and of accessories associated with the gland. An asterisk beside a property name indicates required data.

Thread

Thread type - Select the thread type used for the gland.

Thread size - Select a standard code used to represent the thread size.

Gland seal range

Outer sheath maximum diameter - Type a value for the maximum diameter of the outer gland sheath, including units.

Outer sheath minimum diameter - Type a value for the minimum diameter (after compression) of the outer gland sheath, including units.

Inner sheath maximum diameter - Type a value for the maximum diameter of the inner gland sheath, including units.

Inner sheath minimum diameter - Type a value for the minimum diameter (after compression) of the inner gland sheath, including units.

Components and accessories

Locknut - Select the material of the gland locknut, if appropriate.

IP washer - Select the material of the gland IP washer, if appropriate.

Shroud - Select the material of the gland shroud, if appropriate.

Earth tag - If the gland includes an earth tag, select the material from the list.

Seal - If the gland includes a seal, select the material from the list.

Serrated lock - If the gland includes a serrated lock, select the material from the list.

Armor clamps - Select this check box if the gland is suitable for armored cable.

Related Topics
- *General Tab (Reference Gland Common Properties Dialog Box)*, page 621
- *Procurement Tab (Reference Gland Common Properties Dialog Box)*, page 622
- *Reference Gland Common Properties Dialog Box*, page 620

Procurement Tab (Reference Gland Common Properties Dialog Box)

Allows you to enter information related to ordering glands for your inventory.

Manufacturer - Allows you to select a manufacturer. You can define and customize manufacturers in the Data Dictionary Manager.

Manufacturer part number - Allows you to enter a part number specified by the manufacturer of the gland type.

Stores part number - Allows you to enter a reference number for the gland type in your inventory.

Related Topics
- *Details Tab (Reference Gland Common Properties Dialog Box)*, page 621
- *General Tab (Reference Gland Common Properties Dialog Box)*, page 621
- *Reference Gland Common Properties Dialog Box*, page 620

Reference Cable Library Dialog Box

Enables you to select a reference cable defined in the **Reference Data Explorer** for connecting to a control station.

Reference Cable - Shows the list of reference cables that belong to the cable specification that you select on the **Select Reference Cable** dialog box.

Formation - Describes the structure of the cable in terms of the number of conductors, and where appropriate, additional conductors with their sizes. For example, 3 x 2.5 mm2 + 2 x 1.0 mm2 - that is, three 2.5 mm^2 main conductors and two 2.5 mm^2 additional conductors.

Cable Specification - Indicates the family of cables to which the reference cable belongs corresponding to the cable specification that you select before opening this dialog box. You can define and customize cable specifications in the Data Dictionary Manager

Armor Type - Indicates the type of armor used to protect the cable from its environment. For example, types of armor include non-armored, steel, or lead.

Insulation - Indicates the material used for the cable jacket. For example, insulation materials include PVC and polyethylene.

Temperature - Indicates the manufacturer rating of the maximum operating temperature for the cable.

Voltage Rating - Indicates the standard voltage rating that applies to the cable.

Related Topics
- *Create a Typical Control Station*, page 55

Replace Cables Dialog Box

Replaces the characteristics of a group of project cables using specified properties from a common reference cable. Note that the software does not allow you to replace cables in batch mode if the cables that you want to replace have already been terminated. You can replace a cable manually in the **Common Properties** dialog box if the reference cable has the same conductor arrangement as the cable you want to replace.

Filter criteria

Allows you to select a set of criteria to filter the available reference cables from which you select to replace the existing reference cables.

Cable category - Displays the category that defines the purpose for which the cable is used, that is, power, control, grounding, or instrumentation. The category depends on the folder from which you selected the cables and is read-only.

📝 Note

- The software replaces cables of the same category. For example, you can only replace a power cable with another power cable.

Cable specification - Allows you to select a specification that defines a family of cables, each consisting of a specific group of reference cables.

Conductor arrangement - Allows you to select an option that defines the number and arrangement of the conductors required. Only those reference cables that correspond to a selected conductor arrangement appear. The following standard options are available:

- **Single core cable** - Selects current-carrying cables with a single core.
- **2-core cable** - Selects cables with two current-carrying conductors and no additional conductors.
- **2+1-core cable** - Selects cables with two current-carrying conductors and one additional conductor.
- **3-core cable** - Selects cables with three current-carrying conductors and no additional conductors.
- **3+1-core cable** - Selects cables with three current-carrying conductors and one additional conductor.
- **3+2-core cable** - Selects cables with three current-carrying conductors and two additional conductors.
- **Multicore cable** - Selects cables with multiple conductors.

Size - Allows you to select a predetermined cable size. Only those reference cables that correspond to a selected cable size appear.

Replacement cable

Shows the reference cable and its cable formation. This cable is used to replace the selected cables.

Reference cable - Select the reference cable that you want to use for replacing the selected cables.

Cable formation - Displays the cable formation (number and size of conductors) for the selected reference cable. This property is read-only.

Data window - Displays the cables that you selected in the **Electrical Index**, with details of the cable tag, the cable formation, conductor material, and conductor arrangement. Beside each cable name is a check box that allows you to select the cable for replacing.

Select all - Allows you to select all the cables in the list for replacing.

📝 Notes

- The software replaces cables of the same category. For example, you can only replace a power cable with another power cable.

- The software replaces only the values of the following properties with values from the reference cable that you select:

Cable Category

Cable Specification

Basic Ampacity in Air

Basic Ampacity in Ground

Reactance

Resistivity

Actual Conductor Size

Additional Conductor Size

Air Ampacity Temperature

Armor Type

Bending Radius

Conductor Arrangement

Conductor Material

Conductor Size

Inner Insulation Temperature

Insulation Color

Insulation Material

Insulation Temperature rating

Insulation Thickness

Insulation Voltage Rating

Jacket Color

Jacket Insulation Material

Jacket Insulation Temperature

No of Additional Conductors

No of Conductors

No of Conductors in Group

No of Sets

Outside Diameter

Overall Shield

Pull Tension

Weight per Length

Cable Formation

Electrical Equipment Class

Electrical Equipment Sub Class

Electrical Equipment Type

Stores Part Number

- The software clears the values of the following properties:

Calc Starting Voltage Drop

Calc Voltage Drop Percent

- The software retains the values of all other properties of the selected cables.

- When the selected cables for replacing include parallel cables, only one tag per parallel cable assembly appears in the display; however the software performs the operation for all the parallel cables in each assembly. If descriptions exist for the individual parallel cables, the software retains those descriptions.

- If you want to replace a group of parallel cables generated by the sizing procedure with cables having a different formation, for the **Reference cable** property, you must select **Non-sized power cable**. Otherwise, you can only replace parallel cables with another group of cables that have the same formation. For example, you can only replace three parallel cables of 300 mm^2 (3*300 mm^2) with another configuration of 3 parallel cables, such as 3*90 mm^2, 3*(3*25 mm^2+1*10 mm^2), and so forth.

Related Topics
- *Managing Cables Common Tasks*, page 181
- *Replace Cables*, page 188

Select Reference Cable Dialog Box

Sets a reference cable for the cable drum that you are defining in the **Cable Drum Properties** dialog box.

Filter criteria

Filters the **Reference cable** list, from which you select a reference cable for a given cable drum. Each field for which you select a value further filters the list.

Cable category - From this list, you must select a category that defines the purpose for which the cable is used: **Power**, **Instrumentation**, **Control**, or **Grounding**.

Cable specification - Allows you to select a filter criterion that defines a family of cables, each consisting of a specific group of reference cables.

Conductor arrangement - Allows you to select a filter criterion that defines conductor number and arrangement.

Size - Allows you to select a cable size as a filter criterion.

Select cable

Reference cable - Select a reference cable for the cable drum that you are defining in the **Cable Drum Common Properties** dialog box.

Cable formation - Displays the cable formation (number and size of conductors) for the selected reference cable.

Related Topics
- *Cable Drum Assignment: An Overview*, page 207
- *Cable Drum Common Properties Dialog Box*, page 613
- *Edit Cable Drum Properties*, page 210

Batch Cable Routing Dialog Box

Enables you to define a cable routing path and assign cables to this routing path in batch mode.

Clear routing from selected cables - Select to delete existing routing from selected cables without assigning new cable routing, when you click **Apply** or **OK**. Clear to enable assignment of cables to the routing path that you define in the **Cable routing** group box.

Cable routing

In this group box, you build the routing path to which you then assign cables.

Select predefined route - When you click **Find**, allows you to select a predefined route to use as a basis for the cable routing definition.

Copy route from cable - When you click **Find**, allows you to select routing from another cable to use as a basis for the cable routing definition.

Segregation level - Displays the segregation level that applies for the cables that you selected prior to opening this dialog box. The value of the segregation level is read-only.

Find - opens the **Find** dialog box, from which you can select as predefined route or a cable with routing for copying, according to the option you have selected. Note that only routes or cables that belong to the current segregation level are available for selection.

Save as New Route - Opens the **Route Properties** dialog box, from which you can save the current routing definition as a new predefined route.

Route details

Displays the cableways and segments that make up the routing. Note that only those segments that belong to the current segregation level are available for selection.

Sequence - Indicates the sequence of the segment in the routing path.

Cableway - Select the cableway from which you want to select a segment.

Segment - Select the segment through which you want to route the cables. You can use each segment only once in a route.

Length - Displays the segment length for the selected segment. You define this value in the **Properties** window.

Add - Adds a new segment row below the currently selected row.

Insert - Adds a new segment row above the currently selected row.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order of the segments in the route.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order of the segments in the route.

Remove - Removes the currently selected segment from the route.

Routing length - Displays the total length of the segments that you defined.

Select cables

Displays the cables that you selected in the **Electrical Index** or Tabular Editor, with details of the cable tag, cable formation, cable category, armor type, and routing path. Beside each cable name, in the **Select** column, is a check box that allows you to select the cable for routing.

Select all - Allows you to select all the displayed cables for routing.

Related Topics
- *Batch Route Cables*, page 199
- *Cable Routing: An Overview*, page 195
- *Clear Cable Routing*, page 201
- *Managing Cable Routing Common Tasks*, page 196

Cableway Common Properties Dialog Box

Enables you to set options for defining properties of cableways, including the segments that make up the cableway.

Cableway - Identifies the cableway. The cableway tag appears beside the cableway in the **Electrical Index**. You must enter a value that matches the *naming convention* for the cableway. You define naming conventions in the Options Manager.

Description - Allows you to type a description of the cableway, for information only.

Segments

This section displays details of the segments that make up the cableway. When you start to define the segments, the details appear. You can modify the item tag if required; all other properties are view-only. You can also add segments to a cableway and you can delete and sort segments that do not have cables assigned to them. Note that if you insert a segment in the middle of a cableway that already has assigned cables, the software automatically updates the routing.

Item Tag - Allows you to modify the item tag that identifies the cableway segment.

Reference Component - The reference cableway component used for the segment.

Category - The category to which the segment belongs: **Tray**, **Conduit**, or **Miscellaneous**.

Description - Description of the segment, if applicable.

Add - Adds a new segment row below the currently selected row.

Insert - Adds a new segment row above the currently selected row.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order of the segments in the cableway.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order of the segments in the cableway.

Remove - Removes the currently selected segment from the cableway.

Notes - Allows you to type user notes, for example if you want to provide additional information regarding special features of the cableway.

Cableway Segment Common Properties Dialog Box

Enables you to set options for defining properties of cableway segments used in cable routing. You can specify any one of three categories of cableway segments: trays, conduits, and miscellaneous components. Miscellaneous components are used to make various connections between trays and conduits.

Related Topics
- *Cable Routing: An Overview*, page 195
- *Managing Cable Routing Common Tasks*, page 196

General Tab (Cableway Segment Common Properties Dialog Box)

Allows you to define and edit basic properties of a cableway segment used to route cables.

Item tag - Allows you to specify a tag that uniquely identifies the cableway segment.

Cableway - Displays the cableway with which the segment is associated. This value is read-only.

Description - Allows you to type a description of the cableway segment, for information only.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular segment.

Category - Displays the category of the cableway component used as the reference item for the cableway segment. If no reference cableway component is selected, you can also select a category: **Tray**, **Conduit**, or **Miscellaneous**, which the software uses to filter the results when you click **Find** to look for a reference cableway component to associate.

Segregation level - Allows you to specify a *segregation level* for the segment. All segments that belong to a particular routing path must have the same segregation level, and all cables that you route through those segments must have the same segregation level as those segments.

Reference component

Describes the reference cableway component used as the catalog item for the segment. Default values for some of the segment properties, such as **Length**, **Width**, and some fill properties come from the component.

Find - Opens the **Find** dialog box, where you can select a reference cableway component to which you assign the current cableway segment.

Dissociate - Allows you to clear the association of the selected reference component with the cableway segment.

Length - Allows you to type a value for the length of the segment. You can use the value that you type to represent a space reservation or an actual length of routing.

Diameter - Displays the value of the diameter of the cableway segment. This read-only value comes from the reference cableway component used as the basis for this segment and applies only to segments with category **Conduit**.

Width - Displays the value of the width of the cableway segment. This read-only value comes from the reference cableway component used as the basis for this segment and applies only to segments with category **Tray** or **Miscellaneous**.

Depth - Displays the value of the depth of the cableway segment. This read-only value comes from the reference cableway component used as the basis for this segment and applies only to segments with category **Tray** or **Miscellaneous**.

Notes - Allows you to enter text, for example, if you want to provide additional information about the segment.

Adjacent segments

Displays the names of the segments on either side of the current segment in the cableway. The positions of the segments are determined by the sequence assigned to them on the **Cableway Common Properties** dialog box.

From segment - The segment with sequence number preceding that of the current segment. If no segment precedes the current segment, the software displays **End**.

To segment - The segment with sequence number following that of the current segment. If no segment follows the current segment, the software displays **End**.

Segment fill

The software calculates on the fly the segment fill percent for the cables, These cables are routed on the segment as part of the routing, validating the segment fill percent values against the maximum allowed values that you specify or, as defined by NEC code. The software takes into account the fill factor that you specify and the maximum weight per unit length allowed on the tray. If any of the actual values exceed the allowed values, the software rejects them and displays a warning message. For area or width calculations, the software performs the validation according to the segment category and whether or not the segment is covered. The dynamic text that appears indicates the validation method that the software uses:

Allow overfill - Select this check box to ignore validation of values that exceed the allowed area or weight.

Calculation criteria - Select the desired tray fill calculation (shown in percentage) by **Area** or **Width** or **Auto**. Select **Auto** if you desire the calculation by the **Covered** flag (default). Note that when using **NEC** validation method, the **Calculation criteria** list is disabled.

Maximum area - The maximum available area for the segment. For a covered segment, the software calculates the area by multiplying the width and depth values together. For a conduit, the software calculates the area from the diameter. This value is read-only, and comes from the reference cableway component. Note that When using the **NEC** Validation method, the **Maximum area**, **Allowed area**, **Accumulated area** and **Accumulated diameter** values are calculated in inches.

Allowed area - The actual maximum area of the segment that cables included in the routing are allowed to occupy. The software calculates this value as the product of the maximum area and the fill factor. Note that when using the **NEC** validation method, the **Allowed area** value is derived from **NEC** tables according to the cable type of the cables, located in the tray. Note that when using **NEC** validation method, for conduits, the **Allowed area** value changes according to the number of cables in the conduit. Note that this pane will be activated only if the calculation is by **Area**.

Allowed width - The actual maximum width of the segment that cables included in the routing are allowed to occupy. The software calculates this value as the product of the Maximum width of the tray, the layers and the Fill factor, Using NEC validation method the software derives the Allowed width from the **NEC** tables. Note that this pane will be activated only if the calculation is by width.

Fill factor - Type a value representing the fraction of the maximum area or the maximum width that you want to allow when adding cables to the segment. You can type any positive value, including values that are greater than 1; however, for conduits, covered trays or for NEC, the fill factor value should not exceed 1.

De-rating factor - Used for NEC validation, This code factor can have one of three different values (0.4, 0.5, 0.9), The code multiplies the **Allowable width** or the **Allowable area**.

Accumulated area - The total area occupied by the cables routed through the current segment. When appropriate, the software validates this value against the allowed area value. Note that when using the **NEC** validation method, the **Accumulated area** value may be higher than the **Allowed area** value.

Number of layers - Type an integer indicating the number of layers of cables that you want to allow when the software performs validation by diameter.

Accumulated diameter - The sum of the outer diameters of all the cables routed through the current segment. When appropriate, the software validates this value against the product of the segment width, the fill factor, and the number of layers.

% fill- calculates the values that indicate the segment fill percentage.

Maximum weight - The maximum weight per unit length of cables that the segment is capable of supporting. This value is read-only, and comes from the reference cableway component.

Allowed weight - The actual maximum weight per unit length that cables included in the routing are allowed to reach. The software calculates this value as the product of the maximum weight and the weight factor.

Weight factor - Type a value representing the fraction of the maximum weight per unit length that you want to allow when adding cables to the segment. You can type any positive value, including values that are greater than 1.

Accumulated weight - The total weight per unit length of the cables routed through the current segment.

Related Topics
- *Cable Routing: An Overview*, page 195
- *Managing Cable Routing Common Tasks*, page 196

Associated Cables Tab (Cableway Segment Common Properties Dialog Box)

Displays a list of the cables associated with the segment. The data on this tab is read-only. You make the actual associations on the **Cable Common Properties** dialog box or on the **Batch Route Cables** dialog box.

Item Tag - Identifies the cable. The cable tag appears beside the cable in the **Electrical Index**.

Formation - Displays information based on the total number and the size of the conductors or cable sets specified for the particular cable.

Weight / Length - Displays the weight per unit length of the cable. This value is important when filling the segment to ensure that the total weight per unit length of the cables in the segment does not exceed the maximum allowed value for the segment. This is the value assigned to the **Weight Per Length** property that appears in the **Properties** window.

Outside Diameter - Displays the outside diameter of the cable. When filling a segment, this value is important for validating that the accumulated outside diameter of the cables in the segment does not exceed the maximum allowed value for the segment in the case of an uncovered segment, or that the accumulated area of the cables does not exceed the maximum allowed area in the case of a covered segment. This is the value assigned to the **Outside Diameter** property that appears in the **Properties** window.

Reference Cableway Component Common Properties Dialog Box

Enables you to define reference components for creating cableway segments used in cable routing. You can define three categories of cableway components: trays, conduits, and miscellaneous components. You use miscellaneous components to make various connections between trays and conduits.

General Tab (Reference Cableway Component Common Properties Dialog Box)

Sets options for general properties that identify and characterize the cableway component.

Cableway component tag - Allows you to specify a tag that uniquely identifies the cableway component. This is also the commodity tag that identifies the cableway component as a bulk item.

Category - Shows the purpose for which the cableway component is used. Examples of cableway component categories are tray, conduit, and miscellaneous. The software automatically assigns the category according to the folder in which you create the cableway component, and you cannot change the value.

Description - Allows you to type a description of the cableway component, for information only.

Type - Allows you to select a value for the cableway component type. You can define and customize cableway component types in the Data Dictionary Manager.

Nominal size - Type a numerical value corresponding to the manufacturer's data for the diameter or width of the cableway component. This value is for reference only; it is not used in any software calculations.

Length - specifies the default length for the segments to which you assign this cableway component.

Count quantity - specifies whether to count the tray length Metric or Imperial units (**Length**), or to count the number of items (**Number of Items**).

Covered - Select this check box to specify a covered segment; clear for an uncovered segment. This option applies only to cableway component types for which the category is **Tray** or **Miscellaneous**. Selection of this check box affects the calculation of certain segment fill parameters in the segment that you associate this cableway component with. If you modify this property after assigning a cable to a segment based on this cableway component, the software recalculates the segment fill parameters and accepts the change provided you do not exceed the loading parameter values. Note that the **Covered** calculation applies in case you do not select **Area** or **Width** calculation for the segment.

Ventilated - Select to specify a cableway component that is ventilated. You can use this value for filtering cableway components. Note that when using the **NEC** validation method, the **Ventilated** option is disabled.

Metallic - Select to specify a cableway component that is metallic. You can use this value for filtering cableway components.

Flexible - Applies to conduits only. Select to specify a flexible conduit; clear to specify a rigid conduit. You can use this value for filtering cableway components.

Bending radius - Allows you to specify a bending radius for a component representing a corner or junction segment. For a very wide segment, the value you type is likely to be some average value.

Bottom - Allows you to select a value for the structure of the cableway component bottom; for example, **Ladder**, **Solid**. This option applies only to cableway component types for which the category is **Tray** or **Miscellaneous**. You can define and customize cableway component bottom types in the Data Dictionary Manager. Note that when using **NEC** validation method, the **Bottom** pane does not appear. **Construction** - Specifies the tray type, which affects how the software calculates the segment field. Available values are: **Ventilated channel**, **Solid channel**, **Ladder**, **Ventilated trough**, **Solid bottom tray**. This option appears only when using the **NEC** validation method

Loading parameters

Normal span - The span required between two supports to ensure proper support of the component when fully loaded.

Fill factor - Type a positive value to be used as the default for segments to which you assign this cableway component in your project. The fill factor is a quantity that the software multiplies the maximum diameter or area of the segment by (according to the segment category) in order to calculate the allowed area or diameter for cable fill. Note that when using the **NEC** validation method, the **Fill factor** is limited to one.

Weight factor - Type a positive value to be used as the default for segments to which you assign this cableway component in your project. The weight factor is a quantity that the software multiplies the maximum weight per unit length of the segment by in order to calculate the allowed weight per unit length for cable fill.

Maximum weight per unit length - Allows you to type a value for the maximum weight of cable per unit length that the segment is capable of supporting.

Dimensions

The software uses the values you type for the dimensions to calculate the area of cableway segments that you associate with this cableway component. If you change the value of the dimensions after you have assigned a cable to any of these segments, the software recalculates the segment fill parameters and accepts the change provided you do not exceed the loading parameter values.

Width - Allows you to type a value for the width of the cableway component. This option applies only to cableway components with category **Tray** or **Miscellaneous**.

Depth - Allows you to type a value for the depth of the cableway component. This option applies only to cableway components with category **Tray** or **Miscellaneous**.

Diameter - Allows you to type a value for the diameter of the cableway component. This option applies only to cableway components with category **Conduit**.

Area - Displays a calculated value of the area of the cableway component, according to the width and depth (for trays or miscellaneous components) or according to the diameter (for conduits).

Procurement Tab (Reference Cableway Component Common Properties Dialog Box)

Allows you to enter information related to ordering a specific cableway component for your inventory.

Manufacturer - Allows you to select a manufacturer. You can define and customize manufacturers in the Data Dictionary Manager.

Manufacturer part number - Allows you to enter a part number specified by the manufacturer of the cableway component.

Stores part number - Allows you to enter a reference number for the cableway component in your inventory.

Predefined Routes Dialog Box

Displays all the available routes defined in your project. The information on this dialog box is read-only.

Route list

Route - Displays the name of the route.

Description - Displays a description for the route.

New - Opens the **Route Properties** dialog box to allow you to create a new route.

Properties - Opens the **Route Properties** dialog box to allow you to view or modify the properties of the selected route.

Delete - Deletes the selected route from the database.

Route Properties Dialog Box

This dialog box allows you to display the cableways and segments in a predefined route.

Route - Displays the name of the route.

Description - Displays a description for the route.

Segregation level - You must select a *segregation level* for the route. The software applies the segregation level that you select to all segments in the route. Note that once you have added at least one segment to the routing, you cannot change the segregation level; to do so, you must first remove all the segments.

Route details

Cableway - Select a cableway to filter the segments. You must select a cableway before you can select a segment .

Segment - Select a segment to use in the routing path. Only segments that have the same segregation level as the one defined for the route are available for selection. Note that you can use each segment only once in a route.

Length - Displays the length of the selected segment with the appropriate units. This value is read-only.

Add - Adds a new segment row below the currently selected row.

Insert - Adds a new segment row above the currently selected row.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order of the segments in the route.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order of the segments in the route.

Remove - Removes the currently selected segment from the cableway.

Related Topics
- *Batch Route Cables*, page 199
- *Define Routing for a Single Cable*, page 199
- *Edit Cableway Properties*, page 200
- *Managing Cable Routing Common Tasks*, page 196

New Custom Folder Dialog Box

Allows you to create a custom folder, within an item type folder, in the **Reference Data Explorer** or the **Electrical Index**. You then apply a filter to the custom folder to add the specific items you want to the folder.

Custom folder name — Allows you to type the folders name.

Filter name — Shows the filter name for the custom folder. Click **Browse** to select the desired filter.

Select Filter Dialog Box

Allows you to select an available filter, or add a new filter, that is associated with your custom folder.

Select an available filter by opening a folder in the tree view, selecting the required filter and clicking **OK**.

New — Allows you to add a new filter to the selected filter folder.

Properties — Opens the **Properties** dialog box of the selected filter.

For more information about filters, see Using Filters: An Overview in the *Filter Manager User's Guide*, under *Using Filters: An Overview*.

Equipment Profile Properties Dialog Box

Defines a profile in the **Reference Data Explorer**. For each profile, you can select a typical schematic, a lookup table, a catalog item, associated cables, control stations, and a typical circuit with protection devices. For a motor with an internal heater, you can select a cable and typical circuit for the heater.

Related Topics
- *Create a Profile*, page 61

General Tab (Equipment Profile Properties Dialog Box)

Specifies values for general profile properties that apply to loads in the **Electrical Index**.

Profile tag - Allows you to enter a tag name that identifies the profile.

Item type - Indicates the equipment type according to the folder in which you created the profile. The value is read-only.

Set as default - Defines this profile as the default for the specified equipment type when you apply a profile to a load.

Reference item - Allows you to select a reference item for copying the property values when you apply the profile.

Associated cables

Displays the reference cables that the software associates with the load when you apply the profile.

(Data window) - Describes the properties of the associated cables. In particular, this window shows the conductor arrangement.

Add - Opens the **Select Reference Cable** dialog box to associate a new reference cable with the profile.

Edit - Opens the **Select Reference Cable** dialog box to select a different reference cable for associating with the profile.

Remove - Removes the selected associated cable from the profile.

Typical control station

Displays the names and descriptions of typical control stations that the software associates with the load when you apply the profile.

 Note

- Control stations may have associated cables.

Add - Click to display all the available control stations on the **Find** dialog box, where you can select a control station for the profile.

Remove - Removes the selected control station and its cable from the profile.

Related Topics
- *Miscellaneous Tab (Equipment Profile Properties Dialog Box)*, page 643
- *Protection Devices Tab (Equipment Profile Properties Dialog Box)*, page 641
- *Reference Control Station Common Properties Dialog Box*, page 653
- *Space Heater Tab (Equipment Profile Properties Dialog Box)*, page 642

Protection Devices Tab (Equipment Profile Properties Dialog Box)

Specifies typical circuits for use when applying profiles to equipment in the **Electrical Index**.

Typical circuit - Displays the circuit that the software associates with the equipment when you apply the profile.

Find - Allows you to select a typical circuit.

Clear - Allows you to clear the selected typical circuit association to the equipment.

Function - Indicates the purpose of the selected typical circuit. You define the typical circuit function when you create the typical circuit. For loads, the typical circuit function is always **Feeder**.

Internal components - Displays details of the internal components for the selected typical circuit. For example, an internal component can be a contactor or a fuse.

Related Topics
- *General Tab (Equipment Profile Properties Dialog Box)*, page 641
- *Miscellaneous Tab (Equipment Profile Properties Dialog Box)*, page 643
- *Space Heater Tab (Equipment Profile Properties Dialog Box)*, page 642

Space Heater Tab (Equipment Profile Properties Dialog Box)

Enables you to define cables and typical circuits associated with a space heater. This option is available only for profiles that include a motor with an internal heater.

Associated cables

Displays the reference cables that the software associates with the space heater when you apply the profile.

(Data window) - Describes the properties of the associated cables. In particular, this window shows the conductor arrangement.

Add - Opens the **Select Reference Cable** dialog box to associate a new reference cable with the heater.

Edit - Opens the **Select Reference Cable** dialog box to select a different reference cable for associating with the heater.

Remove - Removes the selected associated cable from the heater.

Typical circuit - Displays the circuit that the software associates with the space heater when you apply the profile.

Find - Click to select a typical circuit.

Clear - Allows you to clear the selected typical circuit association to the space heater.

Function - Indicates the purpose of the typical circuit. You define the typical circuit function when you create the typical circuit. An example of a function is a feeder.

Internal components - Describes details of the internal components for the selected typical circuit.

Related Topics
- *General Tab (Equipment Profile Properties Dialog Box)*, page 641
- *Protection Devices Tab (Equipment Profile Properties Dialog Box)*, page 641

Miscellaneous Tab (Equipment Profile Properties Dialog Box)

Specifies a typical schematic and lookup table for a profile.

Typical schematic - Allows you to select a typical schematic for the item to which you want to apply the profile. The software then uses this typical schematic when you generate a schematic for the item.

Find - Allows you to display all the available typical schematics on the **Find** dialog box, where you can select the desired typical schematic for the profile.

Clear - Allows you to clear the selected typical schematic association to the item.

Lookup table - Allows you to select a lookup table for the desired item to which you apply the profile. The software then uses this lookup table when you generate a schematic for the item.

Find - Allows you to display all the available lookup tables on the **Find** dialog box, where you can select the desired lookup table for the profile.

Clear - Allows you to clear the selected lookup table association to the item.

Typical instruments - Allows you to select one or more typical instruments for the profile. If required, you can add the same typical instrument more than once. When you apply the profile to an item, the software creates associated instruments for the item based on the typical instruments.

Add - Allows you to display all the available typical instruments on the **Find** dialog box, where you can select the desired typical instrument for the profile.

Remove - Removes the currently selected typical instrument from the list.

Related Topics
- *General Tab (Equipment Profile Properties Dialog Box)*, page 641
- *Protection Devices Tab (Equipment Profile Properties Dialog Box)*, page 641
- *Reference Control Station Common Properties Dialog Box*, page 653
- *Space Heater Tab (Equipment Profile Properties Dialog Box)*, page 642

Lookup Table Properties Dialog Box

Defines default values for a set of properties that you can apply to a project item.

Related Topics
- *Create Lookup Tables*, page 62

Data Tab (Lookup Table Properties Dialog Box)

Specifies sets of data values for the properties defined on the **Structure** tab. You usually define the structure once, but you will probably add data quite often. The key values define the uniqueness of each row of data. When you apply a lookup table to a project item and select a particular row, the software enters the values in that row for the appropriate properties of the project item.

Lookup table tag - Allows you to enter a tag name that identifies the lookup table.

Set as default - Select to define this lookup table as the default for the specified equipment type when you apply a lookup table to a load.

📝 **Notes**

- Each time you define a row of data, the software adds a new blank row automatically at the end of the list. To define more than 17 rows, you have to do one of the following: increase the window size, maximize the window, close the window and then re-open it, or press the 'down arrow' key on the keyboard.

- Each combination of data in the key fields must be unique. The software indicates key fields by double sets of asterisks (**) in the header labels. For more details, see *Lookup Table Data Example*, page 67.

- For a 2- or 3-winding transformer, the **Number of Secondaries** property always appears by default as a key field. For a 2-winding transformer, it has a value of 1; for a 3-winding transformer, it has a value of 2.

- You can split the view of the data columns by dragging or double-clicking the split box at the left of the horizontal scroll bar. This allows you to scroll the columns independently in each split region. To remove the split, drag the vertical divider to the left.

- You can sort the rows of data according to a particular column by clicking the column header. Repeatedly clicking a column header displays the data in alternating ascending and descending order.

Remove - Removes the currently selected row of data.

Copy Row - Copies the data from the currently selected row to the Clipboard.

Paste Row - Pastes data from the Clipboard to the currently selected row. This action replaces all existing values in the row.

Excel Report - Click to display the data in this dialog box in a Microsoft Excel file. This is useful if you want to format the layout or print out the data. Also, after correcting errors, you can use the Excel file as a data source for re-importing the data using the Import Manager.

Related Topics
- *Lookup Table Data Example*, page 67
- *Structure Tab (Lookup Table Properties Dialog Box)*, page 645

Structure Tab (Lookup Table Properties Dialog Box)

Specifies the properties for which you want to assign default values to a project item. You define each property as an input (key value) or an output.

Lookup table tag - Allows you to enter a tag name that identifies the lookup table.

Set as default - Select to define this lookup table as the default for the specified equipment type when you apply a lookup table to a project item.

Definition - Displays the properties included in the lookup table.

Add - Adds a new row. You define a property and its characteristics in the **Edit** section of this dialog box.

Remove - Removes the currently selected property from the lookup table definition.

Up - Moves the selected row above the previous row each time that you click this option. The order of the rows affects the order in which the properties appear on the **Data** tab.

Down - Moves the selected row below the next row each time that you click this option. The order of the rows affects the order in which the properties appear on the **Data** tab.

Edit

Provides options for you to select a property and specify how it behaves when you apply the lookup table.

Item type - Displays the item type for which the lookup table applies. The value is read-only, except in the case of transformers, where you can select whether to show the properties of the primary winding or of a secondary winding.

Property - Allows you to select a property for use in the lookup table. In the case of a lookup table for a transformer, the **Number of Secondaries** property appears by default and you cannot delete it.

Caption - Allows you to type any text to use as the caption for the property. The caption appears in the header row on the **Data** tab.

Input / output - Allows you to specify how the software uses the property when you apply the lookup table to a load. Available options are:

- **Input** - The property behaves as a key - that is, it specifies the uniqueness of each set of lookup table values on the **Data** tab, in addition to providing the data value.

- **Output** - The property only provides a data value when you apply the lookup table to a load. This option is disabled for read-only data.

Overwrite option - Allows you to specify whether the property value overwrites an existing value when you apply the lookup table to a load.

Related Topics
- *Data Tab (Lookup Table Properties Dialog Box)*, page 645
- *Lookup Table Structure Example*, page 65

Reference Cable Common Properties Dialog Box

Sets options for defining general cable properties and design details for reference cables from which you create project cables in the **Electrical Index**.

Related Topics
- *Create a Reference Cable*, page 57

General Tab (Reference Cable Common Properties Dialog Box)

Sets options for general properties that identify and characterize the cable. An asterisk beside a property name indicates required data.

Reference cable - Allows you to specify an item tag that uniquely identifies the reference cable. The item tag appears beside the cable in the **Reference Data Explorer**. You must enter a string in this box to define the cable.

Description - Allows you to type a description of the cable, for information only.

Cable category - Defines the purpose for which the cable is used. The cable category can be power, control, grounding, or instrumentation. The software automatically assigns the cable category according to the folder in which you create the cable, and you cannot change the value.

Cable specification - Allows you to select the cable specification, which identifies the cable type and properties, and groups cables that have common properties. You must select a value for this property. You can define and customize cable specifications in the Data Dictionary Manager.

Cable usage - If required, select a value for this property to further categorize the cable, for example, for control voltage definitions, level definitions, and so forth. You can define and customize cable usages in the Data Dictionary Manager.

Armor type - Allows you to specify the type of armor used to protect the cable from its environment. For example, types of armor include non-armored, steel, or lead. You can define and customize armor types in the Data Dictionary Manager.

Outer jacket insulation - Allows you to specify the material used for the cable jacket. For example, insulation materials include PVC and polyethylene. You can define and customize insulation materials in the Data Dictionary Manager.

Notes - Allows you to type user notes, for example if you want to provide additional information regarding special features of the cable.

Construction

Describes the physical characteristics related to the construction of the cable. When the reference item is a non-sized power cable, you can edit the data; otherwise, the items are read-only. The available values depend on the conductor arrangement you select.

Conductor arrangement - Select an option for a new reference cable to define the number and arrangement of the conductors in the cable (not applicable for instrumentation cables). Your selection affects how you can modify the cable construction. The following standard options are available:

- **Single core cable** - Creates a current-carrying cable with a single core. For this option, you can specify an arrangement of any number of single-core conductors, for example, 3 single-core cables, but you cannot specify any additional conductors.

- **2-core cable** - Creates a cable with two current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **2+1-core cable** - Creates a cable with two current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3-core cable** - Creates a cable with three current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **3+1-core cable** - Creates a cable with three current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3+2-core cable** - Creates a cable with three current-carrying conductors and two additional conductors. You cannot modify the cable construction for this option.

- **Multicore cable** - Creates a cable with multiple conductors. For this option, you can specify any number of conductors and additional conductors.

Note

- You can change the names of the standard conductor arrangement items in the Data Dictionary Manager. However, the software performs calculations according to the position in the list of the item that you select. Also, you can define new conductor arrangements that the software can use to filter the available reference cables when sizing project cables.

Set type - For instrumentation cables only, allows you to select a standard value that represents the arrangement of the wires in the set, such as **Wire**, **Pair**, **Pair w/Shield**, **Triad**, **Triad w/Shield**, and so forth. You can select values of this property for a new cable only; you cannot select a different value for an existing or duplicated cable. If the cable sets are not all of the same type, the value **Other** appears.

Cable formation - Displays computed information based on the total number and the size of the conductors or cable sets specified.

Current-carrying conductors - Specifies the number of conductors within the cable used for carrying the current, excluding the neutral, the ground, or additional conductors (not applicable for instrumentation cables). If you selected **Single-core** or **Multicore** as the conductor arrangement, this option allows you to specify the number of conductors that carry the electric current.

Number of sets - For instrumentation cables only, allows you to type the number of cable sets that the cable contains. Click the ellipsis button ⬚ to open the **Cable Set Properties** dialog box where you can modify the set definitions if required.

Size - Allows you to select a standard size for the current-carrying conductors or cable sets. You can define and customize the available cable sizes in the Data Dictionary Manager.

Material - Specifies the conductor material used for the current-carrying conductors or cable sets. The software automatically copies to project power cables the values in the **Conductor Material Coefficients** lookup table that are appropriate for the selected conductor material; these values are required for short circuit cable sizing. You can define and customize conductor materials in the Data Dictionary Manager.

Insulation - Specifies the material used for insulating the current-carrying conductors or cable sets. For example, insulation materials include polyvinyl chloride (PVC) and polyethylene. The software automatically copies to project power cables the values in the **Conductor Insulation Temperatures** lookup table that are appropriate for the selected insulation material; these values are required for short circuit cable sizing. You can define and customize insulation materials in the Data Dictionary Manager.

Additional conductors - Specifies the number of conductors within the cable in addition to the conductors used for carrying the current (not applicable for instrumentation cables). If you selected **Multicore** as the conductor arrangement, this option allows you to specify the number of additional non-current-carrying conductors, such as ground wires.

Overall shield - For instrumentation cables only, select to specify an overall shield for the cable. When you select this option, you need to specify the size and material of the overall shield.

Size - Allows you to select a standard size for the additional conductors or overall shield. You can define and customize the available cable sizes in the Data Dictionary Manager.

Material - For instrumentation cables only, allows you to select the material used for an overall shield. You can define and customize the available materials in the Data Dictionary Manager.

Communication wire - For instrumentation cables only, select to specify a communication wire for the cable.

Color pattern - Select a color pattern for the cable from one of the patterns available. Click the ellipsis button to define or modify a color pattern.

Insulation voltage rating - Allows you to select a standard voltage rating that corresponds to the manufacturer voltage rating for the cable. You can define and customize the available voltage ratings, including units, in the Data Dictionary Manager.

Insulation temperature rating - Allows you to select a standard temperature rating that corresponds to the manufacturer rating of the maximum operating temperature for the cable. You can define and customize the available temperature ratings, including units, in the Data Dictionary Manager.

Basic ampacity in air - Allows you to define the ampere capacity of the cable in air under the basic conditions that the manufacturer specifies. This quantity is a measure of the ability of the cable to deliver electric current in air. Enter a value with the desired units of current. For power cables, this value is required for sizing; for other types of cables, the value is optional.

Basic ampacity in ground - Allows you to define the ampere capacity of the cable in the ground under the basic conditions that the manufacturer specifies. This quantity is a measure of the ability of the cable to deliver electric current under the ground. Enter a value with the desired units of current. For power cables, this value is required for sizing; for other types of cables, the value is optional.

Reactance per unit length - Allows you to enter the reactance per unit length for a cable that carries alternating current (AC) according to the value that the manufacturer specifies for the cable type. Enter a value with the desired units. For power cables, this value is required for sizing; for other types of cables, the value is optional.

Resistance per unit length - Allows you to enter the resistance per unit length according to the value that the manufacturer specifies for the cable type. Enter a value with the desired units. For power cables, this value is required for sizing; for other types of cables, the value is optional.

Parallel cables

Describes the number of cables in parallel and the list of cable names. This information only applies to project cables.

Related Topics
- *Create a Reference Cable*, page 57
- *Design Data Tab (Reference Cable Common Properties Dialog Box)*, page 647

Design Data Tab (Reference Cable Common Properties Dialog Box)

Provides information about the cable drum assignment, routing, length, and so forth. An asterisk beside a property name indicates required data.

Drum assignment

Allows you to select a pulling area and drum properties to use when assigning a project cable.

Drum - This information only applies to project cables.

Pulling area - Select a pulling area to use when routing the cable from the selected drum. You can define and customize pulling areas in the Data Dictionary Manager.

Maximum drum capacity - Type the maximum length of cable that the drum can hold.

Minimum cable length to order - Type the minimum length of cable that you require when ordering new supplies.

Cable routing

This information only applies to project cables.

Design data

Provides information about the physical characteristics of the cable.

Use external routing length - This information only applies to project cables.

Estimated length - Allows you to enter the estimated length of cable before installation for use as a default value in project cables.

Design length - Allows you to enter a more accurate length for the cable after installation for use as a default value in project cables.

Routing length - This information only applies to project cables.

External routing length - This information only applies to project cables.

Tail 1 - Type a value for the spare tail length at one end the cable, if you require, for use as a default value in project cables.

Length - A calculated field that displays the design length, or if not available, the estimated length.

Tail 2 - Type a value for the spare tail length at the other end the cable, if you require, for use as a default value in project cables.

Spare length - Type a value for additional spare cable length, if you require, for use as a default value in project cables.

Actual length - This information only applies to project cables

Related Topics
- *General Tab (Reference Cable Common Properties Dialog Box)*, page 647

Connections Tab (Reference Cable Common Properties Dialog Box)

From / To

Describes information relating to the connection points for the cable. This information only applies to project cables.

Gland - Displays a gland associated with the cable side, if appropriate.

Find - Click to open the **Find** dialog box, where you can select an appropriate cable gland to associate on the **From** or **To** cable side.

Dissociate - Click to dissociate a gland that already has an association with the cable side.

Related Topics
- *General Tab (Reference Cable Common Properties Dialog Box)*, page 647

Procurement Tab (Reference Cable Common Properties Dialog Box)

Enables you to enter information related to ordering a specific cable for your inventory. When the reference item is a non-sized power cable, you can edit the data; otherwise, the items are read-only.

Manufacturer - Allows you to select a manufacturer. You can define and customize manufacturers in the Data Dictionary Manager.

Model number - Allows you to select a cable model. The available models are specific to the manufacturer. You can define and customize models in the Data Dictionary Manager.

Stores part number - Allows you to enter a reference number for the cable in your inventory.

Related Topics
- *General Tab (Reference Cable Common Properties Dialog Box)*, page 647

Reference Control Station Common Properties Dialog Box

Enables you to associate reference cables with a reference control station that you defined in the **Reference Data Explorer**.

Item tag - Allows you to enter a tag name that identifies the control station.

Associated cables

Displays the reference cables that the software associates with the control station.

(Data window) - Describes the properties of the associated cables. In particular, this window shows the conductor arrangement.

Add - Opens the **Select Reference Cable** dialog box to associate a new reference cable with the control station.

Remove - Removes the selected associated cable from the control station.

Typical I/O set

Displays a typical I/O set of signals for the control station.

Find - Allows you to display all the available typical I/O sets on the **Find** dialog box, where you can select the desired typical I/O set for the control station.

Clear - Allows you to clear the association of the selected typical I/O set with the control station.

Associate Reference Cable Dialog Box

Enables you to select an existing reference cable or specify a new reference cable to associate with a control station or an item profile.

Cable specification - Allows you to select an appropriate cable specification. You must select a cable specification before you can select a cable.

Cable definition

Select from cable library - Choose this option to select an existing reference cable.

Reference cable - Indicates the selected reference cable for associating with the control station or profile.

Select - Opens the **Reference Cable Library** window, which shows the cables that belong to the cable specification that you selected.

Specify construction - Select this option to define a new reference cable based on a specified conductor arrangement. This option is available only when you define a reference cable for an item profile and a reference circuit.

Conductor arrangement - Defines the number and arrangement of the conductors in the cable. The following standard options are available:

- **Single core cable** - Creates a current-carrying cable with a single core. For this option, you can specify an arrangement of any number of single-core conductors, for example, 3 single-core cables, but you cannot specify any additional conductors.

- **2-core cable** - Creates a cable with two current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **2+1-core cable** - Creates a cable with two current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3-core cable** - Creates a cable with three current-carrying conductors and no additional conductors. You cannot modify the cable construction for this option.

- **3+1-core cable** - Creates a cable with three current-carrying conductors and one additional conductor. You cannot modify the cable construction for this option.

- **3+2-core cable** - Creates a cable with three current-carrying conductors and two additional conductors. You cannot modify the cable construction for this option.

- **Multicore cable** - Creates a cable with multiple conductors. For this option, you can specify any number of conductors and additional conductors.

Note

- You can change the names of the standard conductor arrangement items in the Data Dictionary Manager. However, the software performs calculations according to the position in the list of the item that you select. For example, a 2-core cable is always the second item in the list.

Conductors - Specifies the number of conductors within the cable used for carrying the current, excluding the neutral, the ground, or additional conductors. If you selected **Single-core** or **Multicore** as the conductor arrangement, this option allows you to specify the number of conductors or cores in the cable that carry the electric current.

Additional conductors - If you selected **Multicore** as the conductor arrangement, this property specifies the number of additional non-current-carrying conductors, such as ground wires.

Related Topics
- *Create a Typical Control Station*, page 55

Schematic Sheet Properties Dialog Box

Allows you to define multiple sheets and sheet properties for a typical schematic drawing.

Sheets

Sheet Number - Displays the sheet number that you typed under **Sheet number** in the **Sheet details** group box.

Description - Displays the sheet description that you typed under **Description** in the **Sheet details** group box.

Add - Adds a new row to allow you to define a new sheet.

Remove - Removes a selected sheet from the list. The software deletes the removed sheet from the database after you click **OK**.

Sheet details

Sheet number - Allows you to type a sheet sequence number for a sheet selected in the **Sheets** list. You will then be able to select this sheet number and assign it to a block when editing the properties of that typical schematic drawing.

Description - Allows you to type a sheet description for a sheet selected in the **Sheets** list.

Typical Circuit Common Properties Dialog Box

Sets options for defining the properties of a typical circuit.

Circuit - Identifies the typical circuit. The item tag appears beside the typical circuit in the **Reference Data Explorer**. Click the ellipsis button to open a dialog box where you can modify the tag segments that are editable.

Circuit type - Displays the type of the typical circuit, which can be **Incomer**, **Feeder**, **Coupler**, or **Bus Riser**.

Details of circuit internals

Displays the components associated with the typical circuit.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Reference Data Explorer**.

Lookup Table - Allows you to select a lookup table that will be associated with the corresponding circuit internal component. You will then be able to apply this associated lookup table to a project circuit when you run the **Apply Lookup Tables** feature on the **Apply Options** dialog box to propagate the lookup table data to the circuit internal components.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Associated cables

Displays the reference cables that the software associates with the typical circuit.

(Data window) - Describes the properties of the associated cables. In particular, this window shows the conductor arrangement.

Add - Opens the **Select Reference Cable** dialog box to associate a new reference cable with the typical circuit.

Remove - Removes the selected associated cable from the typical circuit.

Typical I/O set

Find - Opens the **Find** dialog box, which allows you to select a typical I/O set to associate with the typical circuit.

Clear - Allows you to clear the typical I/O set association to the typical circuit.

Typical Schematic Blocks Dialog Box

Selects blocks for use in the typical schematics that you define in the **Reference Data Explorer**.

Blocks - Displays the list of the blocks in the database.

Add - Adds a new row. You define a block and its characteristics in the **Block details** section of this dialog box.

Delete - Deletes the currently selected block.

Block name - Allows you to enter a name for the block.

Block type - Allows you to select a block type representing the item type associated with the block, for example, a motor.

File name - Displays the name of the drawing file that represents the block graphically. The software supports file types .sym, .dwg, and .igr. Click the ellipsis button ⬚ to navigate to the desired file.

💡 **Tips**

- You can only select drawing files from the default folder specified in the Options Manager; you cannot navigate to files in any other folder.

- You can click **View** to preview the currently selected block before selecting it.

Related Topics
- *Create Typical Schematic Blocks*, page 69

Typical Schematic Common Properties Dialog Box

Defines a typical schematic with associated blocks in the **Reference Data Explorer**. You use the typical schematic as a template to generate a schematic for a specified load in the **Electrical Index**.

Name - Type a name for the typical schematic.

Template - Select a template for the schematic. The template determines the page size and orientation, and the border and title blocks that appear on the schematic.

Typical schematic block properties - Lists the blocks that will comprise the typical schematic.

Add - Adds another row that displays the block properties according to your selections under **Block details**. Make sure that the blocks that you add belong to the same CAD format. Generating a schematic drawing based on a typical schematic that uses different CAD formats (for example, .sym and .dgn files) may result in generation errors.

Remove - Removes the currently selected block from the schematic.

Block name - Select a block from the available blocks in the library.

Block type - Displays the block type representing the item type associated with the block, such as a motor. This value is read-only.

Sequence - This option applies for situations where there is more than one block of the same type in the schematic. Type an integer that represents the sequence in which the software associates the blocks with matching project items that belong to the same item type. The software uses these sequence numbers if there are multiple project items belonging to the same item type, for example, a schematic that contains two control station blocks that have to be associated with project items. The software associates the block that has lowest sequence number with the plant item that has the lowest **Sequence in group** attribute among the project items belonging to same item type. Example: A typical schematic contains two blocks A and B for which sequence numbers are 5 and 8 respectively. If used with a motor that has two control stations CS-1 and CS-2 whose **Sequence in group** attributes are 3 and 8 respectively, the software matches Block A with Control Station CS-1 and Block B with Control Station CS-2.

💡 **Tips**

- You can click **View** to preview the currently selected block before adding it to the typical schematic.

- This option is not available for multi-tag blocks.

Sheet number - Allows you to assign a sheet number to the current block in the typical schematic you are modifying or creating. The values in this list correspond to the sheets that you create in the **Schematic Sheet Properties** dialog box. You can assign the same block to several different sheets if you want the block to appear more than once on different sheets of the same schematic. Note that you should assign a sheet to each block in the typical schematic. Unassigned sheets are not kept in the database by the software. This option is not available or multi-tag blocks.

X - The X-coordinate value, in the CAD application units, of the block insertion point relative to the drawing origin on the left.

Y - The Y-coordinate value, in the CAD application units, of the block insertion point relative to the drawing origin at the bottom.

Multi-Tag Options - Opens the **Multi-Tag Schematic Options** dialog box where you can specify whether to include a tag list on the main drawing page of a multi-item schematic, and the maximum number of rows in the list to allow on the drawing sheets. This option is available when you select a multi-tag block.

Sheets - Opens the **Schematic Sheet Properties** dialog box where you can define multiple sheets for the current typical schematic drawing.

Related Topics
- *Create a Typical Schematic*, page 71

Apply Options Dialog Box

Enables you to apply default profiles, circuits, control stations, schematics, and lookup tables, all of which you define in the **Reference Data Explorer**, to electrical equipment. In this way, you can create items and populate them with data in a batch operation, thus keeping to a minimum the editing that you need to perform for individual items.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143

Typical Circuit Tab (Apply Options Dialog Box)

Sets options for associating typical circuits to electrical equipment.

Enable typical circuit - Select this check box to apply a typical circuit. This check box is used for security to prevent you from applying a typical circuit in error.

Select typical circuit

Clear last applied typical circuit - Clears the typical circuit that you last applied. Select this option if you do not want to apply any typical circuit to the item.

Function - Displays the function of the selected typical circuit. Examples of functions are **Incomer** and **Feeder**.

Typical circuit - Displays the typical circuit for the item. Click **Find** to navigate to the typical circuit that you want to select.

Apply to - Enables you to select the rule that the software implements when associating the typical circuit. The options are:

- **All items** - Applies the typical circuit to all the selected items.

- **Items with no typical circuit** - Applies the typical circuit only to items without an existing typical circuit. Use this option to add a typical circuit only to new items among the items that you selected.

- **Items with a different typical circuit from the selection** - Changes the typical circuit of the selected items to the current typical circuit. Use this option to add a typical circuit to items with a modified typical circuit definition.

- **Items with the same typical circuit as the selection** - Updates an existing typical circuit for the selected items.

📝 Notes

- If you apply a typical circuit to an item that is already associated with a circuit, the software adjusts the components of the circuit to match the typical circuit specified by the item.

- If you modify the components of a typical circuit that already has a reference from an item, the software does not update the change in the reference until you next apply a typical circuit or a profile to the item.

- If the item that you are applying the typical circuit to is connected to a feeder circuit, the system replaces the circuit so that it matches the applied typical circuit as specified in the profile.

- If the item is connected to feeder equipment other than a circuit (such as a variable frequency drive, a transformer, or a battery bank), the software does not apply the typical circuit.

- If more than one item is connected to the same circuit, the software changes the last applied typical circuit for all the items.

- If you apply a typical circuit that includes associated reference cables, the software creates project cables from them and adds the project cables to the circuit with the **To** sides of the cables associated with the circuit.

- You can clear the typical circuit of an item that is not connected to any circuit by selecting **Clear the last applied typical circuit**.

- You can apply a typical circuit on a circuit, provided that the circuit has no equipment connected to it.

Related Topics

Control Stations Tab (Apply Options Dialog Box)

Sets options for applying a control station with a load, converting equipment, generator, battery bank, and disconnect electrical equipment item.

Enable control stations - Select this check box to apply a control station. This check box is used for security to prevent you from applying control stations in error.

(Data window) - Displays the control stations you select for the load. Click **Find** to navigate to the control stations you want to select.

📝 Note

- If you select a reference control station that has one or more associated control cables, the software creates these cables in the control station that you add to the project load.

Apply to - Enables you to select the rule that the software implements when associating the control station. The options are:

- **All items** - Applies the control station to all the selected items. Select **Replace control stations** to replace all existing control stations and their associated control cables; any existing associations, such as a connection of a control cable to a junction box, are retained. Clear **Replace control stations** to add the selected control stations and their associated control cables to the selected loads along with the existing control stations.

- **Items with no control station** - Applies the control station only to items without an existing control station. Use this option to add a control station only to new items among those items that you have selected.

Related Topics
- *Apply a Typical Control Station to an Item*, page 144
- *Equipment Profile Tab (Apply Options Dialog Box)*, page 664
- *Lookup Table Tab (Apply Options Dialog Box)*, page 662
- *Typical Circuit Tab (Apply Options Dialog Box)*, page 660
- *Typical I/O Set Tab (Apply Options Dialog Box)*, page 664
- *Typical Schematic Tab (Apply Options Dialog Box)*, page 665

Lookup Table Tab (Apply Options Dialog Box)

Sets options for applying a lookup table to the selected items.

Enable lookup table - This check box is used for security to prevent you from applying a lookup table in error. You must select this box to apply a lookup table.

Equipment type - Displays the equipment type to which you apply the lookup table corresponding to the item or folder you have selected. Examples of equipment types are motors and cables.

Lookup table - Displays the default lookup table, as defined in the **Lookup Table Properties** dialog box, for the appropriate equipment class. Click **Find** to navigate to the desired lookup table.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143

Lookup Table Tab (Apply Options Dialog Box) - for Circuit Internal Components

Sets options for applying a lookup table to the selected project circuit internal components.

Enable associated lookup tables of circuit internals - This check box is used for security to prevent you from applying a lookup table in error. You must select this box to apply a lookup table.

Equipment type - Displays the equipment type (Circuit) to which you apply the lookup table.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Apply a Typical Control Station to an Item*, page 144
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143

Typical I/O Set Tab (Apply Options Dialog Box)

Sets options for associating a typical I/O set to items in the **Electrical Index**.

Enable typical I/O set - Select this check box to apply a typical I/O set. This check box is used for security to prevent you from applying a typical I/O set in error.

Select typical I/O set

Clear last applied typical I/O set - Clears the typical I/O set that you last applied. Select this option if you do not want to apply any typical I/O set to the item.

Typical I/O set - Displays the typical I/O set for the item. Click **Find** to navigate to the typical I/O set that you want to select.

Apply to - Enables you to select the rule that the software implements when associating the typical I/O set. The options are:

- **All items** - Applies the typical I/O set to all the selected items.

- **Items with no typical I/O set** - Applies the typical I/O set only to items without an existing typical I/O set. Use this option to add a typical I/O set only to new items among the items that you selected.

- **Items with a different typical I/O set from the selection** - Changes the typical I/O set of the selected items to the current typical I/O set. Use this option to add a typical I/O set to items with a modified typical I/O set definition.

Equipment Profile Tab (Apply Options Dialog Box)

Sets options for applying profiles to loads and converting equipment.

Enable equipment profile - Select this check box to apply a profile. This check box is used for security to prevent you from applying a profile in error.

Number of items to create - This option is available only when you select the folder for the item you require prior to selecting **Apply Options**, and after you select **Enable equipment profile**. It allows you to specify the number of new items to create in the **Electrical Index** in accordance with the profile you select.

Equipment type - Displays the equipment type to which you apply the profile corresponding to the item or folder you have selected. Examples of equipment types are motors and cables.

Profile name - Displays the default profile for the appropriate equipment class, as defined in the **Equipment Profile Properties** dialog box. When you select a folder to create new items in batch mode using the profile, the label appears as **Create item based on profile**. Click **Find** to navigate to a different profile.

Apply to - Enables you to select the rule that the software implements when applying the profile. The options are:

- **All items** - Applies the profile to all the selected items.

- **Items with no profile** - Applies the profile only to items without an existing profile. Use this option to add a profile only to new items among the items that you selected.

- **Items with a different profile from the selection** - Changes the profile of the selected items to the current profile. Use this option to add a profile to items with a modified profile definition.

- **Items with the same profile as the selection** - Updates an existing profile for the selected items.

Populate empty item properties - Select this check box to overwrite item properties that contain null data. Clear the check box to update the profile without overwriting data. Whether you select or clear this option, the software does not overwrite existing values.

✎ Notes

- If you modify the components of a typical circuit that already has a reference from an item, the software does not update the change in the reference until you next apply a typical circuit or a profile to the item.

- When an item has associated power cables, and you connect the item to a circuit by applying a profile with a typical circuit, if there is more than one cable, the software associates the free ends of all the power cables with the circuit.

- If you apply a profile for a reference motor with a heater to a project motor without a heater, the software adds a heater to the project motor.

- If the item that you are applying a profile to is connected to a feeder circuit, the system replaces the circuit so that it matches the applied typical circuit as specified in the profile.

- If the item is connected to feeder equipment other than a circuit (such as a variable frequency drive, a transformer, or a battery bank), the software does not apply the profile.

Related Topics

Typical Schematic Tab (Apply Options Dialog Box)

Sets options for associating typical schematics to loads.

Enable typical schematic - Select this check box to apply a typical schematic. This check box is used for security to prevent you from applying a typical schematic in error.

Select typical schematic

Clear last applied typical schematic - Clears the typical schematic that you last applied. Select this option if you do not want to apply any typical schematic to the load.

Typical schematic - Displays the typical schematic for the load. Click **Find** to navigate to the typical schematic that you want to select.

Apply to - Enables you to select the rule that the software implements when associating the typical schematic. The options are:

- **All items** - Applies the typical schematic to all the selected loads.

- **Items with no typical schematic** - Applies the typical schematic only to loads without an existing typical schematic. Use this option to add a typical schematic only to new loads among the loads that you selected.

- **Items with a different typical schematic from the selection** - Changes the typical schematic of the selected loads to the current typical schematic. Use this option to add a typical schematic to loads with a modified typical schematic definition.

Related Topics
- *Apply a Typical Schematic to an Item*, page 146
- *Control Stations Tab (Apply Options Dialog Box)*, page 661
- *Equipment Profile Tab (Apply Options Dialog Box)*, page 664
- *Lookup Table Tab (Apply Options Dialog Box)*, page 662
- *Typical Circuit Tab (Apply Options Dialog Box)*, page 660
- *Typical I/O Set Tab (Apply Options Dialog Box)*, page 664

Batch Load Association Dialog Box

Enables you to balance load flow of a particular bus and associate multiple loads with a power distribution board (PDB) and a bus. Also, allows you to display the calculated consumption for the entire power distribution board or a particular bus belonging to the selected power distribution board. You can also assign a load to a particular phase or define a load as a three-phase load.

Power distribution board - Allows you to select a power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus to narrow down the load association.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Loads with a PDB in Batch Mode*, page 219

Bus Tab (Batch Load Association Dialog Box)

Allows you to specify the rated data for a selected bus.

Power distribution board - Allows you to select a power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus to narrow down the load association.

Bus rated data

Line voltage - Allows you to specify a voltage associated with the bus line.

Maximum line current - Allows you to specify the maximum value of the line current.

Maximum active power - Allows you to specify the maximum kW value for the selected bus.

Maximum reactive power - Allows you to specify the maximum kVAr value for the selected bus.

Maximum apparent power - Allows you to specify the maximum kVA value for the selected bus.

Supply - Displays the supply (AC or DC) of the selected bus. You define these values in the **Properties** window of the current bus.

Phase - Displays the phase number for the selected bus. (Applies to AC current supply only.)

Overload percentage - Allows you to specify the overload percentage on the maximum line current. The line current determines the maximum allowed load on the bus. The overload percentage can have a value between 0 and 100. For example, if you specify a maximum line current of 100 A and an overload percentage value of 20, the software determines that the bus is overloaded if the total calculated line current exceeds 120 A. You can view overload data for a particular bus on the **Total Bus Load Validation** dialog box.

Spare load percentage - Allows you to specify a value between 0 and 100 for the spare capacity on the bus for future expansion. The software uses this value to calculate the available spare capacity of the bus.

Required power factor - Allows you to specify a power factor value that is required for the current bus. When calculating the electrical consumption for this bus, the software also calculates the power factor of a capacitor bank that is needed to correct the power factor of this bus. For details, see *Correcting Bus Power Factors Using Capacitor Banks*, page 242.

Loads Tab (Batch Load Association Dialog Box)

Enables you to balance the load flow of a particular bus and associate multiple loads with a power distribution board (PDB) and a selected bus. Also, lets you display the calculated consumption for a particular bus belonging to the selected power distribution board. You can also assign a load to a particular phase or define a load as a three-phase load.

Power distribution board - Specifies the power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus to narrow down the load association.

Bus phases

The check boxes in this group box allow you to filter the display of loads in the **Associated loads** data window according to their assigned phases. You define the bus phase labels in the bus **Properties** window.

Phase 1 - Filters the **Associated loads** list and displays only the loads that are associated with phase 1.

Phase 2 - Filters the **Associated loads** list and displays only the loads that are associated with phase 2.

Phase 3 - Filters the **Associated loads** list and displays only the loads that are associated with phase 3.

All phases - Displays all the loads disregarding their phase association.

Create circuits and cells

Create circuits - Enables you to automatically create a feeder circuit for all the loads that are not associated with circuits. This selection does not affect the associated loads that are already associated with feeder circuits. The software creates a circuit based on the last applied typical circuit. If no typical circuit is associated with the current load, the software creates a feeder circuit without any circuit components.

Create cells - Enables you to automatically create a cell where the software creates the feeder circuits for the associated loads.

Associated loads

Displays the loads or converting electrical equipment that you added to be associated with the current PDB and bus. Also, displays the loads or converting electrical equipment that you have already associated with the selected power distribution board before opening this dialog box. This data window displays the key properties of the loads, including load type, consumed and rated power, full load current, and details of their electrical connection hierarchy. Also, allows you to assign a load to a particular phase or define it at as a multi-phase load by selecting all three phases. All of the displayed values apart from phase assignment are read-only.

Item Tag - Displays the item tag of the selected load.

Type - Displays the electrical equipment sub-class of the load, for example, motor, heater.

Consumed Active - Displays the consumed kW of the selected load.

Consumed Reactive - Displays the consumed kVar of the selected load.

Rated Active - Displays the rated kW of the selected load.

Rated Reactive - Displays the rated kVAr of the selected load.

Full Load Current - Displays the full load current for the selected load.

Circuit - Displays the circuit with which the load is associated.

Cell - Displays the cell with which the selected load is associated.

Phase 1 - Allows you assign the selected load to phase 1.

Phase 2 - Allows you assign the selected load to phase 2.

Phase 3 - Allows you assign the selected load to phase 3.

Move to Cache - Unassigns the currently selected load from its associated bus and moves it to the **Cached Loads** tab for reassignment.

Add - Adds a load or converting electrical equipment to be associated with the selected power distribution board.

Remove - Cancels the association of the highlighted load or converting electrical equipment from the PDB or bus. You cannot remove a load that is already connected to a circuit.

Add Selected Loads - Adds the loads you selected in the list view pane of the **Electrical Index**. These loads become associated with the current PDB after you click **OK** or **Apply**.

Calculated results (consumed values)

Displays the calculated consumed values for all the associated loads, such as motors, heaters, converting electrical equipment (for example, a VFD) and so forth for a selected bus filtered according to phase definition.

Electrical consumption using coincidence factors - Displays calculated consumed values using coincidence factors for either direct loads or all feeders.

For direct loads only - Calculates the total consumed power only for those loads that are connected directly to the current bus or through a feeder circuit. All other loads are considered indirect loads including all types of converting equipment.

For all feeders - Calculates the total consumed power for all the loads fed by the current bus as well as the loads that are fed by the buses that are connected to the current bus (as specified in Options Manager). When you select this option, the **Circuits** tab becomes available.

Active [kW] - Displays the calculated total consumed active (at the demand point) electrical power for all the associated loads in kilowatts.

Reactive [kVAR] - Displays the calculated total consumed reactive electrical power for all the associated loads in kilovolt amperes reactive. When displaying the calculated consumed reactive electrical power for all the associated loads in kilovolt amperes and associating a capacitor, the software subtracts the rated kVAR value of that capacitor from the total connected rated kVAR value. Therefore, the consumed electrical power (kVAR) may display a negative value. This means that there is a lagging power factor.

Apparent [kVA] - Displays the calculated total apparent consumed electrical power for all the associated loads in kilovolt amperes.

Required capacitor correction [kVAR] - Displays the calculated value for the power factor of the capacitor that is needed to correct the bus power factor. For details, see *Correcting Bus Power Factors Using Capacitor Banks*, page 242.

Line current [A] - Displays the calculated line current for the associated loads. The software calculates this value based on the total running load.

Power factor - Calculates the power factor by dividing the total rated active power (in kilowatts) by the total apparent rated power (in kilovolt amperes). If there is a lagging power factor, this value may be negative. The software calculates this value based on the total running load.

Notes - Displays an appropriate message regarding the associated loads. For example, **Incomplete load data**.

Note

- You can set a preference that instructs the software to account for transformer power losses. For details, see *Customize All Feeder Load Summary Report Preferences*, page 116.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Batch Load Association Dialog Box*, page 666

Unassigned Loads Tab (Batch Load Association Dialog Box)

Allows you to associate unassigned loads that are associated with a PDB but have not yet been associated with a bus.

Power distribution board - Specifies the power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus with which you want to associate the selected item tags.

Item Tag - Displays the item tag for a load that has not been associated with a bus.

Type - Displays the electrical equipment sub-class of the load, for example, motor, heater.

Select - Selects the item tag to be assigned to the current bus.

Assign To Bus - Associates the selected item tags with the current bus after clicking **Apply**.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Batch Load Association Dialog Box*, page 666

Circuits Tab (Batch Load Association Dialog Box)

Allows you to display all the circuits that are connected to the current bus and which feed other buses.

Power distribution board - Specifies the power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus with which you want to associate the selected item tags.

Item Tag - Displays the item tag for the circuit connected to the current bus and which feeds other circuits.

Description - Displays the circuit description.

Circuit Type - Displays the circuit type, for example: Feeder, Incomer.

Rated Active Power - Displays the rated kW of the circuit.

Rated Reactive Power - Displays the rated kVAr of the circuit.

Rated Apparent Power - Displays the rated kVA of the circuit.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Batch Load Association Dialog Box*, page 666

Cached Loads Tab (Batch Load Association Dialog Box)

Displays a list of loads that you have unassigned from buses in order to reassign them to a different bus for the purpose of load balancing.

Power distribution board - Specifies the power distribution board with which you associate the loads.

Bus - Allows you to select a specific bus to which you want to re-associate the loads in the cache.

Item Tag - Displays the load item tag.

Type - Displays the electrical equipment sub-class of the load, for example, motor, heater.

Consumed Active - Displays the consumed active power consumption for the load.

Consumed Reactive - Displays the consumed reactive power consumption for the load.

Rated Active - Displays the rated active power consumption for the load.

Rated Reactive - Displays the rated reactive power consumption for the load.

Full Load Current - Displays the manufacturer's stated full load current for the load.

Assign To Bus - Assigns the cached loads to the current bus.

Related Topics
- *Associate a Single Load with a PDB*, page 221
- *Associate Loads with a PDB in Batch Mode*, page 219
- *Batch Load Association Dialog Box*, page 666

Battery Bank Common Properties Dialog Box

Sets options for defining the general properties of a battery bank. Also, this dialog box defines properties that the software uses for calculating electrical output and associating the battery bank with the electrical supply.

General Tab (Battery Bank Common Properties Dialog Box)

Sets options for general properties that identify and characterize the battery bank. An asterisk beside a property name indicates required data.

Item tag - Identifies the battery bank. The tag is the name that appears beside the battery bank in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the battery bank. Click the ellipsis button ⊡ to open a dialog box where you can modify the item tag segments that are editable.

✎ **Note**

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the battery bank, for information only.

Manufacturer - Allows you to select a manufacturer for the battery bank. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a battery bank model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular battery bank.

Type - Allows you to enter a description of the type of battery bank such as **Lead-acid** or **Nicad**. The software uses this description in filters and reports. You can define and customize types in the Data Dictionary Manager.

Last applied profile - Indicates the last profile that you applied to the battery bank, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the battery bank, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the battery bank, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the battery bank. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the battery bank. You can define and customize mountings in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the battery bank.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143
- *Controls and Signals Tab (Battery Bank Common Properties Dialog Box)*, page 676
- *Electrical Data Tab (Battery Bank Common Properties Dialog Box)*, page 674

Electrical Data Tab (Battery Bank Common Properties Dialog Box)

Provides information about the battery bank properties based on the output power requirements.

Rated active power - Shows the maximum quantity of electrical power, usually in kW, that the battery bank is able to supply to the network.

Rated voltage - Select the voltage which the battery bank is designed to supply. This quantity usually appears on the battery bank nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on the **Electrical Data** tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the battery bank supplies when operating at maximum power. The full load current usually appears on the nameplate of the battery bank. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually. When the battery bank has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Flooded seal - Select whether the battery bank seal type is flooded or not flooded.

Number of cells - Type the number of individual cells that make up the battery bank.

Float voltage - Allows you to select a battery float voltage from a standard set of values. The float voltage is the voltage that has to be applied to the battery to maintain its charge level.

Related Topics
- *Controls and Signals Tab (Battery Bank Common Properties Dialog Box),* page 676
- *Electrical Calculations*, page 531
- *General Tab (Battery Bank Common Properties Dialog Box),* page 674
- *Instruments Tab (Battery Bank Common Properties Dialog Box)*, page 677

Controls and Signals Tab (Battery Bank Common Properties Dialog Box)

Provides optional information for control stations that control the battery bank operation, and controls and signals for the battery bank.

Associated control stations

Displays a list of control stations associated with the battery bank. This information is read-only and appears if you associated a control station with the battery bank from the **Control Station Common Properties** dialog box.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the item where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be the battery bank, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the battery bank, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the battery bank.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Battery Bank Common Properties Dialog Box)*, page 674
- *General Tab (Battery Bank Common Properties Dialog Box)*, page 674

Instruments Tab (Battery Bank Common Properties Dialog Box)

Displays instruments that monitor and control the operation of the battery bank.

Associated instruments

Displays a list of instruments associated with the battery bank. This information is read-only and appears if you associated an instrument with the battery bank from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Related Topics
- *Controls and Signals Tab (Battery Bank Common Properties Dialog Box)*, page 676
- *Electrical Data Tab (Battery Bank Common Properties Dialog Box)*, page 674
- *General Tab (Battery Bank Common Properties Dialog Box)*, page 674

Converting Equipment Common Properties Dialog Box

Sets options for defining the general properties of converting equipment such as a transformer. Also, this dialog box defines properties that the software uses for calculating electrical loads and associating the equipment with the electrical supply.

Related Topics
- *Create a Project Item*, page 53
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38

General Tab (Converting Equipment Common Properties Dialog Box)

Sets options for general properties that identify and characterize the converting equipment. An asterisk beside a property name indicates required data.

Consumer - You can use this property for filtering items that you want to appear in the Tabular Editor or in reports. The software does not select this check box by default for converting equipment.

Item tag - Identifies the item of equipment. The tag is the name that appears beside the item in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the item. Click the ellipsis button to open a dialog box where you can modify the item tag segments that are editable.

Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the item, for information only.

Manufacturer - Allows you to select a manufacturer for the converting equipment. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select an item model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Process equipment - Allows you to enter a description of the equipment that the converting equipment affects or is associated with.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular load.

Converting equipment type - Displays the electrical equipment sub-class of the converting equipment, for example, transformer or UPS. The value is read-only. The software uses this description in filters and reports.

Type - Allows you to enter a description of the specific type of converting equipment indicated under **Converting equipment type**. The software uses this description in filters and reports. You can define and customize types in the Data Dictionary Manager.

Last applied profile - Indicates the last profile that you applied to the equipment, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the equipment, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the equipment, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the converting equipment. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the type of equipment. You can define and customize mountings in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the item.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143
- *Controls and Signals Tab (Converting Equipment Common Properties Dialog Box)*, page 681
- *Electrical Data Tab (Converting Equipment Common Properties Dialog Box)*, page 679
- *Feeder Data Tab (Converting Equipment Common Properties Dialog Box)*, page 684
- *Instruments Tab (Converting Equipment Common Properties Dialog Box)*, page 683

Electrical Data Tab (Converting Equipment Common Properties Dialog Box)

Provides information about the converting equipment properties based on the output power requirements.

Rated power - Allows you to enter the desired maximum input power for the equipment. This quantity usually appears on the equipment nameplate. You must enter a value with appropriate units of measure for the standard that you are using. The units of measure depend on the type of equipment. The software specifies the units of measure in the Data Dictionary Manager.

Consumed power - Allows you to enter the electrical power losses of the converting equipment, including appropriate units such as kilowatts.

Rated voltage - Select the input voltage at which the equipment is designed to operate. This quantity ensures electrical compatibility between the electrical supply and the equipment. It usually appears on the equipment nameplate.

Supply - Select the type of supply: alternating current (AC) or direct current (DC).

Frequency (Hz) - Select the operating frequency of the equipment. This property applies for an AC supply only.

Number of phases - Select the number of phases in the power supply to the equipment. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Phase arrangement - Select the desired value to determine which of the available phases are in use for the current equipment, for example 3 PH + N (three phases plus neutral). This property applies for an AC supply only and the values on this list that are available for selection depend on the number of phases that you have selected.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on the **Load Data** tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the equipment draws from the supply when operating at maximum power. The full load current usually appears on the equipment nameplate. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually.

Resistive losses - Allows you to enter a value for the active power loss that is caused by the resistance of the windings (copper losses). The software will use this value to account for converting equipment power losses when calculating batch load assignment and generating an All Feeder Load Summary report. The software will account for these losses if you set your report preferences accordingly. (Click **File** > **Preferences** > **Reports** and then select the **Account for converting equipment power losses**.)

Reactive losses - Allows to enter a value for the reactive power loss that is caused by the magnetic effects in the core (iron losses). The software will use this value to account for converting equipment power losses when calculating batch load assignment and generating an All Feeder Load Summary report. The software will account for these losses if you set your report preferences accordingly. (Click **File** > **Preferences** > **Reports** and then select the **Account for converting equipment power losses**.)

Secondary / output / (S-10)

This section provides information about the output from the equipment. When the converting equipment is a transformer, the software displays the item tags for the secondary windings, as well as appropriate information according to whether the transformer design has one or two sets of secondary windings. Note that this section is available for all types of converting equipment apart from limiting current reactors.

Rated power - Allows you to enter the desired maximum output power for the equipment. The total rated power for the output (or for all sets of secondary windings in the case of a transformer) cannot exceed the total rated power for the input.

Nominal current - Enter the output current for the converting equipment, including appropriate units such as amperes. For a transformer, this is the current in the secondary windings.

Rated voltage - Select the output voltage at which the equipment is designed to operate. This quantity ensures electrical compatibility between the electrical supply and the equipment. It usually appears on the equipment nameplate.

Current limiting reactors

This section enables you to define the reduction of the short-circuit power of networks or installations to an acceptable value with regard to the short-circuit strength of the equipment or the breaking capacity of the circuit breaker.

Insulation level - Allows you to enter the rated voltage level for the insulation of the equipment.

Resistance - Allows you to enter the resistive value of the current limiting reactor.

Reactance - Allows you to enter the reactance values of the current limiting reactor.

Short circuit current [RMS] - Allows you to enter the level of fault current (Root Mean Square).

Short circuit current [Peak] - Allows you to enter the maximum level of the fault current.

Power factor - Allows you to enter an appropriate power factor for the current limiting reactor.

Basic impulse level - Allows you to define the maximum voltage level for the impulse test of the current limiting reactor.

Controls and Signals Tab (Converting Equipment Common Properties Dialog Box)

Provides optional information for control stations that control the equipment operation, and controls and signals for the equipment.

Associated control stations

Displays a list of control stations associated with the equipment. This information is read-only and appears either if you associated a control station with the equipment from the **Control Station Common Properties** dialog box.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the item where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be the converting equipment, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the equipment, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the equipment.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Converting Equipment Common Properties Dialog Box)*, page 679

Instruments Tab (Converting Equipment Common Properties Dialog Box)

Displays instruments that monitor and control the operation of the equipment.

Associated instruments

Displays a list of instruments associated with the equipment. This information is read-only and appears if you associated an instrument with the equipment from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Related Topics
- *Controls and Signals Tab (Converting Equipment Common Properties Dialog Box)*, page 681
- *Electrical Data Tab (Converting Equipment Common Properties Dialog Box)*, page 679
- *Feeder Data Tab (Converting Equipment Common Properties Dialog Box)*, page 684
- *General Tab (Converting Equipment Common Properties Dialog Box)*, page 678

Feeder Data Tab (Converting Equipment Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in items of equipment that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

✎ Note

- Once you associate the equipment with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the feeder for the converting equipment item. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

✎ Note

- Where the feeder source is an item of equipment other than a PDB, the source tag appears in the **Feeder equipment** box and all the properties in this section are disabled.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the equipment, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the equipment is assigned. The values in this list depend on the selected power distribution board. If you select a bus without specifying a circuit, the software assigns the bus to the equipment, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display the cell/bucket/compartment to which the equipment is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the equipment is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

✎ Note

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the equipment is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the equipment, the software assigns circuit components based on that typical circuit.

Note

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the equipment-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the equipment is assigned. The software automatically assigns the cell name.

Phase 1 - Select to associate the current equipment with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current equipment with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current equipment with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

Note

- To update the circuit components, you need to modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Feeder equipment

Where equipment other than a PDB circuit feeds the converting equipment, the software displays the item tag of the feeding equipment in this box. For example, if the feeder equipment is a generator, the generator tag is displayed.

🖉 Note

- Where a feeder equipment tag appears, the properties under the **Location** section are disabled. Likewise, where the feeder source is a PDB, the source data appears under the **Location** section and this property is disabled.

Last applied typical circuit

Indicates the last typical circuit that you applied to the converting equipment, if appropriate.

Tag - Displays the tag of the last applied typical circuit.

Function - Displays the function of the last applied typical circuit. For a converting equipment item, this is always **Feeder**.

Related Topics
- *Controls and Signals Tab (Converting Equipment Common Properties Dialog Box)*, page 681
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Converting Equipment Common Properties Dialog Box)*, page 679
- *General Tab (Converting Equipment Common Properties Dialog Box)*, page 678
- *Instruments Tab (Converting Equipment Common Properties Dialog Box)*, page 683

Alternative Feeder Tab (Converting Equipment Common Properties Dialog Box)

Displays the location of an alternative feeder, if defined, in items of equipment that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the equipment is assigned, if defined.

Circuit - Displays the circuit to which the equipment is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics
- *Controls and Signals Tab (Converting Equipment Common Properties Dialog Box)*, page 681
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Converting Equipment Common Properties Dialog Box)*, page 679
- *Feeder Data Tab (Converting Equipment Common Properties Dialog Box)*, page 684
- *General Tab (Converting Equipment Common Properties Dialog Box)*, page 678
- *Instruments Tab (Converting Equipment Common Properties Dialog Box)*, page 683

Copy Driver Power Absorbed Dialog Box

This dialog box allows you to copy the power values for selected motors from the **Driver Power Absorbed** property of the item of process equipment to the **Brake Power** property of the motor with which it is associated. This functionality exists for process equipment and motors that have been retrieved from SmartPlant P&ID.

Select - Allows you to select the row for copying the values when you click **Apply** or **OK**.

Process Equipment Tag - Identifies the process equipment item.

Description - Displays the description of the process equipment item, if it exists.

PID Drawing Number - Displays the PID drawing number of the process equipment item.

Driver Rated Power - Displays the driver rated power of the process equipment item.

Driver Power Absorbed - Displays the driver power absorbed of the process equipment item. This is the property that is copied when you select the item.

Electrical Equipment Tag - Displays the tag of the electrical equipment, such as a motor, with which the process equipment item is associated.

Motor Rated Power - Displays the value of the motor rated power, if the process equipment item is associated with a motor.

Brake Power - Displays the current value of the brake power of the motor, before copying the values.

Heater Rated Power - Displays the value of the heater rated power, if the process equipment item is associated with a heater.

Select all - Selects all of the process equipment items for copying data.

Description Dialog Box

This dialog box allows you to enter a long text value in the **Description** text box in the **Common Properties** dialog box for an item. For long descriptions, click the ellipsis button [...] to bring up the **Description** dialog box.

Description - Allows you to enter a description for the item.

Note

- Very long descriptions cannot be displayed in their entirety in labels, reports, and so forth.

Related Topics
- *Review and Edit Item Properties*, page 110

Disconnect Electrical Equipment Common Properties Dialog Box

Sets options for defining the general properties of disconnect electrical equipment.

Item tag - Identifies the disconnect equipment. The item tag appears beside the disconnect equipment in the **Electrical Index**. The item tag matches the *naming convention* for the disconnect equipment. Click the ellipsis button [...] to open a dialog box where you can modify the tag segments that are editable.

Type - Displays the disconnect equipment type.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the equipment, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the equipment.

Last applied typical schematic - Indicates the last typical schematic that you applied to the equipment, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the equipment, if appropriate. This information is read-only.

Associated control stations

Displays a list of control stations associated with the equipment. This information is read-only and appears if you associated a control station with the load from the **Control Station Common Properties** dialog box.

Item Tag - Displays the tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Displays a list of all the I/O signals associated with the equipment. This information is read-only and appears if you associated an I/O signal with the disconnect equipment.

Item Tag - Displays the tag for the associated I/O signal.

Equipment Type - Displays the disconnect equipment type.

Description - Displays the description for the I/O signal, if entered.

I/O Type - Displays the I/O type defined for the signal, such as AO, AI, Fieldbus, and so forth.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143

Electrical Motor Common Properties Dialog Box

Sets options for defining the general properties of an electrical motor. Also, this dialog box defines properties that the software uses for calculating loads and associating the motor with the electrical supply.

Related Topics
- *Create an Electrical Motor*, page 128
- *Create Load Items in Batch Mode*, page 131
- *Duplicate an Electrical Motor*, page 130
- *Edit Electrical Motor Common Properties*, page 129

General Tab (Electrical Motor Common Properties Dialog Box)

Sets options for general motor properties that identify and characterize the motor. An asterisk beside a property name indicates required data.

Consumer - You can use this property for filtering items that you want to appear in the Tabular Editor or in reports. The software selects this check box by default when you create a new motor.

Item tag - Identifies the motor. The tag is the name that appears beside the motor in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the motor. Click the ellipsis button ⊡ to open a dialog box where you can modify the item tag segments that are editable.

📝 Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the motor, for information only. Click the ellipsis button ⊡ to open a dialog box where you can type longer text for the description.

📝 Note

- When typing the description in the dialog box, to begin a new line, press **Ctrl** + **Enter**.

Manufacturer - Allows you to select a manufacturer for the motor. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a motor model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Process equipment - Allows you to enter a description of the equipment that the motor drives. If the motor was retrieved from SmartPlant P&ID and included an association with an item of process equipment, the software fills this property with the name of the process equipment.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular motor.

Conformity to standard - Allows you to specify the standard for electrical calculations when you associate a power cable with the motor. The standard affects the sizing calculation that the software uses for an associated cable. The standard options defined in the Data Dictionary Manager are:

- **National Electrical Code (NEC)** - The standard used in the US. When you select the NEC standard, the software allows you to use the NEC lookup tables to automate population of motor data in your project.

- **International Electrotechnical Commission (IEC)** - The standard used primarily in Europe.

The conformity to standard also determines the available values of conductor size, frequency, and rated voltage.

Motor design letter - Allows you to select a design letter, which is a standard designation for NEC motors. The design letter provides an indication of the values of various mechanical and electrical characteristics of the motor.

Type - Allows you to enter a description of the type of motor. The software uses this description in filters and reports. You can define and customize types in the Data Dictionary Manager.

Last applied profile - Indicates the last profile that you applied to the motor, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the motor, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the motor, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the motor. For example, standards include **Totally Enclosed Air Cooled (TEAC)** and **Totally Enclosed Fan Cooled (TEFC)**. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the type of the load or machine. You can define and customize mountings in the Data Dictionary Manager.

Process operational requirements - Allows you to select the operational requirements of the plant in which the motor is located. The following standard system options are available:

- Emergency

- Normal plant operation

- Shutdown

- Startup

You can define additional values for this list in the Data Dictionary Manager.

Frame size - Provides a list of US standard values that define the physical and mechanical properties and dimensions of the motor. You can define and customize frame sizes in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the motor.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143
- *Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)*, page 698
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *Feeder Data Tab (Electrical Motor Common Properties Dialog Box)*, page 702
- *Instruments Tab (Electrical Motor Common Properties Dialog Box)*, page 700
- *Load Data Tab (Electrical Motor Common Properties Dialog Box)*, page 692

Load Data Tab (Electrical Motor Common Properties Dialog Box)

Provides information about the mechanical power and usage requirements of the motor. You can find many of these quantities on the motor nameplate. An asterisk beside a property name indicates required data.

Rated power - Allows you to select a standard rated power value that represents the maximum mechanical power that a motor shaft is required to supply. This quantity usually appears on the motor nameplate. You can define and customize the rated power values in the Data Dictionary Manager. The available rated power values can only have hp or kW as the units of measure.

Brake power - The minimum mechanical power required to make the motor shaft rotate and drive the pump or process equipment attached to the motor. If you enter a value, it cannot be greater than the value of the rated power and must have appropriate units of measure for the standard that you are using: W, kW, MW, or hp. You can specify the default project units of measure in the Data Dictionary Manager.

Minimum required power - Allows you to enter a value that corresponds to a minimum power requirement of the motor. This quantity is applicable if you are using the IEC (European) standard. If you enter a value, it cannot be greater than the value of the rated power and must have appropriate units of measure: W, kW, MW, or hp. You can specify the default project units of measure in the Data Dictionary Manager.

Supply - Select the type of supply: alternating current (AC) or direct current (DC).

Number of poles - Select the number of poles for the motor. For example, you can select 2 or 4 poles. You can define and customize the available number of poles in the Data Dictionary Manager.

Rated voltage - Select the voltage at which the motor is designed to operate. This quantity usually appears on the motor nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Particular power - Allows you to enter the power requirement of the motor under a specific operating point that the user specifies. This quantity is applicable if you are using the IEC standard. If you enter a value, it cannot be greater than the value of the rated power and must have appropriate units of measure: W, kW, MW, or hp. You can specify the default project units of measure in the Data Dictionary Manager.

Number of phases - Select the number of phases in the power supply to the motor. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Frequency (Hz) - Select the operating frequency of the motor. You can define and customize the available frequencies in the Data Dictionary Manager. This property applies for an AC supply only.

Phase arrangement - Select the desired value to determine which of the available phases are in use for the current load, for example 2 PH + N (two phases plus neutral). This property applies for an AC supply only and the values on this list that are available for selection depend on the number of phases that you have selected.

Coincidence factors

Allows you to enter values corresponding to the percentage utilization of the motor for the selected operating mode. Also known as duty factors. Type a value between 0 and 1 for the desired mode. A value of 0 corresponds to an unused motor, while a value of 1 corresponds to continuous operation.

Operating mode - Select the operating mode that describes how the motor is used in practice in the plant. The standard available operating modes are:

- Continuous
- Intermittent
- Standby
- Spare

Note

- The coincidence factor corresponding to the selected operating mode provides a quantitative measure of the motor behavior in that mode. You can customize the available operating modes in the Data Dictionary Manager.

Operating factor - Enter a value between 0 and 1 to represent the ratio of the load power requirement to the rated power at an *operating point* that you specify. The software multiplies the consumed electrical power by the operating factor to calculate average power usage.

Demand factor - Displays a read-only value that represents the ratio of the brake power to the rated power. The software calculates the demand factor automatically using the values of the brake power and the rated power that you specify on this dialog box tab. The demand factor influences the consumed electrical power for the motor.

LRC power factor - Allows you to enter the value of the ratio of the *active power* to the *apparent power* at locked rotor current (LRC). The LRC power factor can have a value between 0 and 1.

Number of wires - Select the number of wires used to supply power to the motor.

Synchronous speed - Displays the maximum theoretical rate of rotation of a machine shaft, in revolutions per minute (rpm), assuming no energy losses due to the magnetic field, pole architecture, and so forth.

Asynchronous speed - Allows you to enter the actual rate of rotation of a motor shaft. If you enter a value, it cannot be greater than the value of the synchronous speed.

Te stall time - Allows you to enter the time that it takes the motor to come to rest after you switch off the power.

Run-up time - Allows you to enter the time that it takes the motor to reach its operating speed after you switch on the power.

Power factor and efficiency

Enter the values under the conditions of 50%, 75%, and 100% of the rated power. The software uses these values with the rated power and demand factor to calculate the power factor and efficiency at the brake power.

Power factor - Enter the ratio of *active power* to *apparent power* under the conditions of 50%, 75%, and 100% of the rated power. The power factor can have a value between 0 and 1.

Efficiency - Enter the ratio of power output to power input under the conditions of 50%, 75%, and 100% of the rated power. The efficiency can have a value between 0 and 1.

At demand - The software calculates the power factor and efficiency at the brake power by linear interpolation of the values you enter at 50%, 75%, and 100% of the rated power.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Schematic to an Item*, page 146
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)*, page 698
- *Electrical Calculations*, page 531
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *Feeder Data Tab (Electrical Motor Common Properties Dialog Box)*, page 702
- *General Tab (Electrical Motor Common Properties Dialog Box)*, page 691
- *Instruments Tab (Electrical Motor Common Properties Dialog Box)*, page 700

Electrical Data Tab (Electrical Motor Common Properties Dialog Box)

Provides information about the electrical motor properties based on the mechanical power requirements.

Electrical power

The software calculates the electrical power based on load data for the motor. The values are all read-only. The software can perform the calculation at three operating points:

- **Rated** - Maximum power.
- **Consumed** - Normal operation at the motor brake power.
- **Particular** - An additional point that you can specify.

Active - Shows the quantity of electrical power in kW drawn from the network and supplied to the motor that is actually converted to mechanical work.

Reactive - Shows the quantity of reactive power in kVAR that the motor draws from the network.

Apparent - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This quantity is the actual power that the network needs to generate.

Average - Shows the average electrical power consumption for the motor. The software uses the average electrical power consumption to calculate overall electrical load requirements. The software calculates the average load by multiplying the consumed active electrical power by the *operating factor* that you entered on the **Load Data** tab.

Full load current and locked rotor current data

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. The following options are available:

- **Calculated** - The software calculates the full load current value based on values you enter on the **Load Data** tab (rated power and rated voltage); the calculated value overwrites any previous value. You can then type a value manually for the **Locked rotor current** or the **LRC to FLA ratio** property; whichever one of these values you type, the software then calculates the remaining value automatically.

- **Manual** - Enables data entry in the **Full load current** box. In this mode, you must first type a value for the full load current. Next, you can type a value for the **Locked rotor current** or the **LRC to FLA ratio** property; whichever one of these values you type, the software then calculates the remaining value automatically.

- • **Tables** - Enables you to use NEC or other lookup tables to apply values for the **Full load current** and **Locked rotor current** properties (in this mode, these properties and the **LRC to FLA ratio** property are read-only). With this option, you can select **Use locked rotor indicating code**, in which case the software calculates a value for the **Locked rotor current** property by using the maximum value for the specified indicating code letter. If you clear **Use locked rotor indicating code**, the software applies a value from the lookup tables (if applied). Note that to make this option available, you must first import the NEC lookup tables into your plant (for details, see the Options Manager User's Guide: *Import Reference Data*) and on the **General** tab, you must select **NEC** as the value for **Conformity to standard**.

Full load current (FLA) - The electrical current, in amperes, that the motor draws from the supply when operating at maximum power. The full load current usually appears on the motor nameplate. You can modify the value of this property only when you select **Manual** from the **Override FLA calculation** list. Note that in the **Calculated** mode, the actual value calculated depends on whether the motor is single- or 3-phase. When the motor has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Locked rotor current (LRC) - Shows the initial electrical current, in amperes or kilo amperes, required to start rotation of the motor. The software calculates this value automatically when values for the full load current and the LRC / FLA ratio already appear. Note that in **Manual** mode, if you want to enter a value in this field manually so that the software can calculate the value of the LRC / FLA ratio, you must first type a value for the **Full load current** property. When you select **Tables** as the value of the **Override FLA calculation** property, you can apply a value from the tables or select an indicating code letter for the software to calculate a value.

LRC to FLA ratio - Shows the value of the ratio of locked rotor current to full load current, usually shown on the motor nameplate. The software calculates this value automatically when values for the full load current and the locked rotor current already appear. Note that in **Manual** mode, if you want to enter a value in this field manually so that the software can calculate the value of the locked rotor current, you must first type a value for the **Full load current** property.

Use locked rotor indicating code - Select to enable the **Indicating code letter (kVA / HP)** list and then select a code letter from the list. Some manufacturers indicate the code letter on the motor nameplate. This option is only available when you select the **Tables** mode.

Indicating code letter (kVA / HP) - Select the code that specifies the power requirement for the motor with locked rotor, applicable if you are working with the NEC standard. Each code letter represents a specified range of kilovolt-amperes per horsepower with locked rotor. The software uses the highest value in the range to calculate the locked rotor current.

Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)

Provides optional information for control stations that control the motor operation, and controls and signals for the motor.

Associated control stations

Displays a list of control stations associated with the motor. This information is read-only and appears either if you associated a control station with the motor from the **Control Station Common Properties** dialog box or if you applied a profile defined in the **Reference Data Explorer**.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the motor where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be the motor, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the motor, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the motor.

Controller

Provides controller details in the case where there is a device that controls the motor operation.

Type of controller - Describes the controller mechanism. Examples of controller mechanisms are solid- state relay (SSR), silicon controlled rectifier (SCR), and electronic.

Variable frequency drive - Select this check box if the controller is a variable-frequency drive.

Restart options

Auto restart - Select the desired option to indicate whether motor restart after a power outage is automatic or manual.

Restart group - Select the group of motors to which the motor belongs when restarting. After a power outage, the motors restart in groups, rather than all together, to avoid overloading the network.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *Feeder Data Tab (Electrical Motor Common Properties Dialog Box)*, page 702
- *General Tab (Electrical Motor Common Properties Dialog Box)*, page 691
- *Instruments Tab (Electrical Motor Common Properties Dialog Box)*, page 700
- *Load Data Tab (Electrical Motor Common Properties Dialog Box)*, page 692

Instruments Tab (Electrical Motor Common Properties Dialog Box)

Displays instruments that monitor and control the motor operation.

Associated instruments

Displays a list of instruments associated with the motor. This information is read-only and appears if you associated an instrument with the motor from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Related Topics
- *Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)*, page 698
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *Feeder Data Tab (Electrical Motor Common Properties Dialog Box)*, page 702
- *General Tab (Electrical Motor Common Properties Dialog Box)*, page 691
- *Load Data Tab (Electrical Motor Common Properties Dialog Box)*, page 692

Feeder Data Tab (Electrical Motor Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in motors that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

Note

- Once you associate the motor with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the motor feeder. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

Note

- Where the feeder source is an item of equipment other than a PDB, the source tag appears in the **Feeder equipment** box and all the properties in this section are disabled.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the motor, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the motor is assigned. The values in this list depend on the selected power distribution board. If you select a bus without specifying a circuit, the software assigns the bus to the motor, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display a cell/bucket/ compartment to which the motor is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the motor is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

Note

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the motor is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the motor, the software assigns circuit components based on that typical circuit.

Note

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the motor-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the motor is assigned. The software automatically assigns the cell name.

Note

- If you select both the **Create cell** and **Create circuit** check boxes, the software creates a circuit directly under the cell in the **Electrical Index**.

Phase 1 - Select to associate the current load with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current load with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current load with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

Note

- To update and replace the circuit components, you need to re-apply a typical circuit to the motor or modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Feeder equipment

Where equipment other than a PDB circuit feeds the motor, the software displays the item tag of the feeding equipment in this box. For example, if a Variable Frequency Drive (VFD) feeds the motor, the VFD tag is displayed.

📝 Note

- Where a feeder equipment tag appears, the properties under the **Location** section are disabled. Likewise, where the feeder source is a PDB, the source data appears under the **Location** section and this property is disabled.

Last applied typical circuit

Indicates the last typical circuit that you applied to the motor, if appropriate.

Tag - Displays the tag of the last applied typical circuit.

Function - Displays the function of the last applied typical circuit. For a motor, this property appears as **Feeder**.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)*, page 698
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *General Tab (Electrical Motor Common Properties Dialog Box)*, page 691
- *Instruments Tab (Electrical Motor Common Properties Dialog Box)*, page 700
- *Load Data Tab (Electrical Motor Common Properties Dialog Box)*, page 692

Alternative Feeder Tab (Electrical Motor Common Properties Dialog Box)

Displays the location of an alternative motor feeder, if defined, in motors that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the motor is assigned, if defined.

Circuit - Displays the circuit to which the motor is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Circuit to an Item*, page 143
- *Controls and Signals Tab (Electrical Motor Common Properties Dialog Box)*, page 698
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Electrical Motor Common Properties Dialog Box)*, page 695
- *Feeder Data Tab (Electrical Motor Common Properties Dialog Box)*, page 702
- *General Tab (Electrical Motor Common Properties Dialog Box)*, page 691
- *Instruments Tab (Electrical Motor Common Properties Dialog Box)*, page 700
- *Load Data Tab (Electrical Motor Common Properties Dialog Box)*, page 692

Generator Common Properties Dialog Box

Sets options for defining the general properties of a generator. Also, this dialog box defines properties that the software uses for calculating electrical output and associating the generator with the electrical supply.

Related Topics
- *Create a Project Item*, page 53
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38

General Tab (Generator Common Properties Dialog Box)

Sets options for general properties that identify and characterize the generator. An asterisk beside a property name indicates required data.

Item tag - Identifies the generator. The tag is the name that appears beside the generator in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the generator. Click the ellipsis button [...] to open a dialog box where you can modify the item tag segments that are editable.

📝 Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the generator, for information only.

Manufacturer - Allows you to select a manufacturer for the generator. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a generator model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Process equipment - Allows you to enter a description of the equipment that the generator affects.

Exciter type - Allows you to select the type of exciter used to establish the magnetic field in the stator; for example, **Fixed**, **Brushless**, **Static**. The software uses this description in filters and reports. You can define and customize exciter types in the Data Dictionary Manager.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular generator.

Type - Allows you to enter a description of the type of generator; for example **Turbo**, **Hydro**, **Air-cooled**. The software uses this description in filters and reports. You can define and customize types in the Data Dictionary Manager.

Last applied profile - Indicates the last profile that you applied to the generator, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the generator, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the generator, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the generator. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the type of the generator. You can define and customize mountings in the Data Dictionary Manager.

Governor type - Allows you to select the type of governor used to control the generator speed; for example, **Steam turbine**, **Gas turbine**, **Digital**. The software uses this description in filters and reports. You can define and customize governor types in the Data Dictionary Manager.

Frame size - Provides a list of US standard values that define the physical and mechanical properties and dimensions of the generator. You can define and customize frame sizes in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the generator.

Related Topics

Electrical Data Tab (Generator Common Properties Dialog Box)

Provides information about the load properties based on the output power requirements.

Rated apparent power - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This is the quantity of mechanical power in kW that the generator requires to supply maximum electrical power to the network.

Power factor - Enter the ratio of *active power* to *apparent power* at 100% of the rated power and at the *demand point*. The power factor can have a value between 0 and 1.

Rated active power - Shows the maximum quantity of electrical power in kW that the generator needs to supply to the network.

Supply - Select the type of supply that the generator produces: alternating current (AC) or direct current (DC).

Number of phases - Select the number of phases in the generator output. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Frequency (Hz) - Select the operating frequency of the generator. You can define and customize the available frequencies in the Data Dictionary Manager. This property applies for an AC output only.

Rated voltage - Select the voltage which the generator is designed to supply. This quantity usually appears on the generator nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Efficiency - Enter the ratio of power output to power input under the conditions of 50%, 75%, and 100% of the rated power. The efficiency can have a value between 0 and 1.

Number of poles - Select the number of poles for the generator. For example, you can select 2 or 4 poles. You can define and customize the available number of poles in the Data Dictionary Manager.

Synchronous speed - Displays the maximum theoretical rate of rotation of a machine shaft, in revolutions per minute (rpm), assuming no energy losses due to the magnetic field, pole architecture, and so forth.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select

Calculated, the software calculates the full load current value based on values you enter on this tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the generator supplies when operating at maximum power. The full load current usually appears on the nameplate of the generator. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually.

Control mode - Allows you to select the generator control mode that determines how the generator behaves under different load conditions; for example, **Isochronous**, **Droop**, **Isochronous/Droop**. The software uses this description in filters and reports. You can define and customize control modes in the Data Dictionary Manager.

Related Topics
- *Controls and Signals Tab (Generator Common Properties Dialog Box)*, page 709
- *Electrical Calculations*, page 531
- *General Tab (Generator Common Properties Dialog Box)*, page 707
- *Instruments Tab (Generator Common Properties Dialog Box)*, page 710

Controls and Signals Tab (Generator Common Properties Dialog Box)

Provides optional information for control stations that control the generator operation, and controls and signals for the generator.

Associated control stations

Displays a list of control stations associated with the generator. This information is read-only and appears if you associated a control station with the generator from the **Control Station Common Properties** dialog box.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the item where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be the generator, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the generator, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the generator.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Generator Common Properties Dialog Box)*, page 707
- *General Tab (Generator Common Properties Dialog Box)*, page 707
- *Instruments Tab (Generator Common Properties Dialog Box)*, page 710

Instruments Tab (Generator Common Properties Dialog Box)

Displays instruments that monitor and control the operation of the generator.

Associated instruments

Displays a list of instruments associated with the generator. This information is read-only and appears if you associated an instrument with the generator from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Instrument Common Properties Dialog Box

Sets options for defining the general properties of instruments. Also, this dialog box defines properties that the software uses for calculating electrical loads and associating the instrument with the electrical supply.

Related Topics
- *Create a Project Item*, page 53
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38

General Tab (Instrument Common Properties Dialog Box)

Sets options for general properties that identify and characterize an instrument. An asterisk beside a property name indicates required data.

Item tag - Identifies the instrument. The tag is the name that appears beside the instrument in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the instrument. Click the ellipsis button ▭ to open a dialog box where you can modify the item tag segments that are editable.

✎ Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the instrument, for information only.

Manufacturer - Allows you to select a manufacturer for the instrument. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a instrument model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular instrument.

Type - Allows you to select an instrument type such as **I/P TRANSDUCER** or **D/P TYPE FLOW TRANSMITTER**. The available instrument types depend on the selected process function. The software uses this value in filters and reports and when specifying data for import or export.

Process function - Allows you to select a process function such as flow, level, relief valve, and so forth.

Criticality - Allows you to select the desired instrument criticality, which is used for classifying instruments for maintenance purposes. The software uses this value in filters and reports and when specifying data for import or export.

System I/O type - Allows you to select an instrument system I/O type such as **AO** (analog output) or **PI** (pulse input).

Reference loop number - Allows you to type the number of a control loop to which the instrument belongs, if appropriate.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the instrument, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** to the instrument.

Notes - Allows you to enter text, for example, if you want to provide additional information about the instrument.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Associated Items Tab (Instrument Common Properties Dialog Box)*, page 714
- *Electrical Data Tab (Instrument Common Properties Dialog Box)*, page 712
- *Feeder Data Tab (Instrument Common Properties Dialog Box)*, page 717
- *Instrument Data Tab (Instrument Common Properties Dialog Box)*, page 716

Electrical Data Tab (Instrument Common Properties Dialog Box)

Provides information about instrument properties and properties of the equipment associated with the instrument.

Supply - Select the type of supply that the instrument requires: alternating current (AC) or direct current (DC).

Number of phases - Select the number of phases for the instrument. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Frequency (Hz) - Select the operating frequency of the instrument. You can define and customize the available frequencies in the Data Dictionary Manager. This property applies for an AC output only.

Instrument rated power - Allows you to enter the power requirement for the instrument.

Power factor - Enter the ratio of *active power* to *apparent power* at 100% of the rated power and at the *demand point for the associated equipment*. The power factor can have a value between 0 and 1.

Rated voltage - Select the voltage at which the instrument is designed to operate. This quantity usually appears on the instrument nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Rated active power - Shows the quantity of electrical power in kW drawn from the network and supplied to the associated equipment that is actually converted to output power.

Rated reactive power - Shows the quantity of reactive power in kVAR that the associated equipment draws from the network.

Rated apparent power - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This quantity is the actual power that the network needs to generate for the associated equipment.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on this tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the instrument measures for the associated equipment when operating at maximum power. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually. When the associated equipment has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Starting current - The electrical current, in amperes, that the instrument measures for the associated equipment when starting. When the instrument has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Coincidence factors

Allows you to enter values corresponding to the percentage utilization of the associated equipment for the selected operating mode. Also known as duty factors. Type a value between 0 and 1 for the desired mode. A value of 0 corresponds to an unused load, while a value of 1 corresponds to continuous operation.

Operating mode - Select the operating mode that describes how the associated equipment is used in practice in the plant. The standard available operating modes are:

- Continuous
- Intermittent
- Standby
- Spare

Note

- The coincidence factor corresponding to the selected operating mode provides a quantitative measure of the associated equipment behavior in that mode. You can customize the available operating modes in the Data Dictionary Manager.

Related Topics
- *Associated Items Tab (Instrument Common Properties Dialog Box)*, page 714
- *Feeder Data Tab (Instrument Common Properties Dialog Box)*, page 717
- *General Tab (Instrument Common Properties Dialog Box)*, page 712
- *Instrument Data Tab (Instrument Common Properties Dialog Box)*, page 716

Associated Items Tab (Instrument Common Properties Dialog Box)

Provides information for the equipment and cables that are associated with the instrument.

Associated equipment - Displays the item of electrical equipment associated with the instrument.

Note

- You can associate one electrical item only per instrument.

Find - Opens the **Find** dialog box, which allows you to select an electrical item to associate with the instrument.

Clear - Allows you to clear the electrical item association to the instrument.

Associated cables

Displays the project cables that the software associates with the instrument.

(Data window) - Describes the names and cable specifications of the associated cables.

Add - Opens the **Find** dialog box, which allows you to select control, grounding, or instrumentation cables to associate with the instrument.

Note

- You can associate more than one cable per instrument.

Remove - Removes the selected associated cable from the instrument.

Related Topics

Instrument Data Tab (Instrument Common Properties Dialog Box)

Displays instrument data including switching properties and measurement ranges.

Output type - The output types provides an indication of how the instrument output behaves in response to the input. Output can be linear or square root, for example. The value you select is informative only and does not affect any other data values. You can define and customize output types in the Data Dictionary Manager.

Switch action - Where the instrument includes a mechanical or an electronic switch, select an option that describes how the switch behaves, for example, **Energize to trip**, **Open on failure**, and so forth.

Solenoid supply - Where the instrument includes a solenoid, select the type of supply to the solenoid: alternating current (AC) or direct current (DC).

Solenoid voltage - Where the instrument includes a solenoid, select the voltage at which the solenoid is designed to operate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Solenoid action - Where the instrument includes a solenoid, select an option that describes how the solenoid behaves, for example, **Energize to trip**, **Open on failure**, and so forth.

Set point 1 value - Type a value for a set point (input) at which the instrument alarms or trips, if required, and select the appropriate units from the list.

Set point 2 value - Type a value for a second set point (input) at which the instrument alarms or trips, if required, and select the appropriate units from the list.

Set point 3 value - Type a value for a third set point (input) at which the instrument alarms or trips, if required, and select the appropriate units from the list.

Set point 4 value - Type a value for a fourth set point (input) at which the instrument alarms or trips, if required, and select the appropriate units from the list.

Lower range value - Type a value for the lowest input that the instrument can accept and select the appropriate units from the list.

Higher range value - Type a value for the highest input that the instrument can accept and select the appropriate units from the list.

Minimum output value - Type a value for the lowest output that the instrument is required to produce and select the appropriate units from the list.

Maximum output value - Type a value for the highest output that the instrument is required to produce and select the appropriate units from the list.

Display scale minimum - Type a value for the minimum value that normally appears on the instrument display and select the appropriate units from the list. This is usually important for analog displays.

Display scale maximum - Type a value for the maximum value that normally appears on the instrument display and select the appropriate units from the list.

Related Topics
- *Associated Items Tab (Instrument Common Properties Dialog Box)*, page 714
- *Electrical Data Tab (Instrument Common Properties Dialog Box)*, page 712
- *Feeder Data Tab (Instrument Common Properties Dialog Box)*, page 717
- *General Tab (Instrument Common Properties Dialog Box)*, page 712

Feeder Data Tab (Instrument Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in instruments that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

Note

- Once you associate the instrument with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the instrument feeder. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the instrument, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the instrument is assigned. The values in this list depend on the selected power distribution board. If you select a bus without specifying a circuit, the software assigns the bus to the instrument, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display a cell/bucket/ compartment to which the instrument is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the instrument is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

📝 **Note**

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the instrument is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the instrument, the software assigns circuit components based on that typical circuit.

📝 **Note**

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the instrument-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the instrument is assigned. The software automatically assigns the cell name.

Phase 1 - Select to associate the current instrument with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current instrument with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current instrument with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Electrical Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

Note

- To update the circuit components, you need to modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Alternative Feeder Tab (Instrument Common Properties Dialog Box)

Displays the location of an alternative feeder, if defined, in instruments that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the instrument is assigned, if defined.

Circuit - Displays the circuit to which the instrument is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics

Item Tag Dialog Box

The item tag dialog box appears whenever you create, or duplicate an item in the **Reference Data Explorer** or **Electrical Index** or whenever you duplicate an item.

In **Common Properties** dialog boxes, the **Item Tag** dialog box opens when you click the ellipsis button ⸬ beside the **Item tag** text box.

Item tag - Allows you to type text for the item tag, which can either be free text or a mask that fits the item naming convention according to whether you invoke the use of the naming convention for the item type. You define and activate the naming convention for the item type in Options Manager.

Metering Equipment Common Properties Dialog Box

Enables you to specify general and electrical data for equipment used to measure electrical parameters.

General Tab (Metering Equipment Common Properties Dialog Box)

Sets options for general properties that identify and characterize the metering equipment. An asterisk beside a property name indicates required data.

Item tag - Identifies the metering equipment. The tag is the name that appears beside the metering equipment in the **Electrical Index**. The item tag matches the *naming convention* for the metering equipment. Click the ellipsis button ⸬ to open a dialog box where you can modify the item tag segments that are editable.

Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the metering equipment, for information only.

Manufacturer - Allows you to select a manufacturer for the metering equipment. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a metering equipment model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Type - Displays the type of metering equipment, such as **Potential Transformer**, **Ammeter**, or **Multimeter**. The metering equipment type determines the electrical properties that appear on this dialog box. This value is read-only.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular metering equipment item.

Design type - Allows you to select a standard that specifies the construction of the metering equipment. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the metering equipment. You can define and customize mountings in the Data Dictionary Manager.

Primary rating - Enter the voltage applied on the primary winding. Applies to current transformers and potential transformers only.

Secondary rating - Enter the voltage output on the secondary winding. Applies to current transformers and potential transformers only.

Ratio - The calculated ratio of the primary rating to the secondary rating. Applies to current transformers and potential transformers only.

Connection type - Allows you to specify whether a potential transformer is used to measure voltages between two lines or between one line and ground.

Operating voltage - Select a nominal operating voltage for a voltmeter.

Operating current - Enter a nominal operating current for an ammeter, including units. Note that the software does not validate this value against the minimum and maximum meter ranges.

Minimum meter range - Enter the minimum voltage or current that the meter is designed to measure, including units. Applies to ammeters and voltmeters only.

Maximum meter range - Enter the maximum voltage or current that the meter is designed to measure, including units. Applies to ammeters and voltmeters only.

Notes - Allows you to enter text, for example, if you want to provide additional information about the metering equipment.

Ranges Tab (Metering Equipment Common Properties Dialog Box)

Allows you to set nominal operating values and ranges for specified properties of a multimeter.

Current

Select if using the meter to measure current values.

Operating current - Enter a nominal operating current for the meter, including units. Note that the software does not validate this value against the minimum and maximum meter ranges.

Minimum meter range - Enter the minimum current that the meter is designed to measure, including units.

Maximum meter range - Enter the maximum current that the meter is designed to measure, including units.

Voltage

Select if using the meter to measure voltage values.

Operating voltage - Select a nominal operating voltage for the meter.

Minimum meter range - Enter the minimum voltage that the meter is designed to measure, including units.

Maximum meter range - Enter the maximum voltage that the meter is designed to measure, including units.

Active power

Select if using the meter to measure active power values.

Operating power - Enter a nominal operating value of active power for the meter, including units. Note that the software does not validate this value against the minimum and maximum meter ranges.

Minimum meter range - Enter the minimum active power that the meter is designed to measure, including units.

Maximum meter range - Enter the maximum active power that the meter is designed to measure, including units.

Reactive power

Select if using the meter to measure reactive power values.

Operating power - Enter a nominal operating value of reactive power for the meter, including units. Note that the software does not validate this value against the minimum and maximum meter ranges.

Minimum meter range - Enter the minimum reactive power that the meter is designed to measure, including units.

Maximum meter range - Enter the maximum reactive power that the meter is designed to measure, including units.

Frequency

Select if using the meter to measure frequency values.

Operating frequency - Select a nominal operating frequency for the meter.

Minimum meter range - Enter the minimum frequency that the meter is designed to measure, including units.

Maximum meter range - Enter the maximum frequency that the meter is designed to measure, including units.

Power factor

Select if using the meter to measure power factor values.

Operating PF - Type a nominal operating power factor value for the meter.

Minimum meter range - Type the minimum power factor that the meter is designed to measure.

Maximum meter range - Type the maximum power factor that the meter is designed to measure.

Related Topics
- *Create Meters*, page 259

Functions Tab (Metering Equipment Common Properties Dialog Box)

Allows you to specify functions for a protection relay.

Functions

Displays the relay functions according to the data based on the reference function that you select with default or modified values as specified under **Function details**. The values are all read-only.

Add - Adds another row and opens the **Find** dialog box which allows you to select a reference function as a basis for the relay function you define.

Remove - Removes the currently selected function.

Function details

After you select a reference function, this section allows you to modify the values. The software retains any values that you modify for the specific protection relay.

Name - Select a function from the list of available functions.

Code - Type a relay function code value if you require.

Setting (%) - Allows you to specify a value representing the percentage of the entire measurement range at which the relay action takes place.

Setting - Allows you to specify the absolute value of the measurement at which the relay action takes place.

Relay action - Select the action for the relay to perform at the specified setting value.

Related Topics
- *Create Protection Relays*, page 261
- *Create Relay Functions*, page 260

Other Electrical Equipment Common Properties Dialog Box

Sets options for defining the general properties of an item of equipment such as a lighting fixture or a socket outlet. Also, this dialog box defines properties that the software uses for calculating electrical loads and associating the item with the electrical supply.

Related Topics
- *Create a Project Item*, page 53
- *Create Miscellaneous Electrical Equipment*, page 133
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38

General Tab (Other Electrical Equipment Common Properties Dialog Box)

Sets options for general properties that identify and characterize the load. An asterisk beside a property name indicates required data.

Consumer - You can use this property for filtering items that you want to appear in the Tabular Editor or in reports. The software selects this check box by default when you create a new item.

Item tag - Identifies the load. The tag is the name that appears beside the load in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the tag matches the *naming convention* for the load. Click the ellipsis button to open a dialog box where you can modify the item tag segments that are editable.

📝 Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the load, for information only.

Manufacturer - Allows you to select a manufacturer for the load. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a load model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Process equipment - Allows you to enter a description of the equipment that the load affects.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular load.

Type - Allows you to select one of the following electrical equipment sub-classes: **Lighting Fixture**, **Socket Outlet**, **Welding Outlet**, or **Special Electrical Equipment**. When you select a value, the software moves the item to the appropriate sub-folder of the Electrical Index. The software uses this description in filters and reports.

Last applied profile - Indicates the last profile that you applied to the load, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the load, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the load, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the load. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the type of the load or machine. You can define and customize mountings in the Data Dictionary Manager.

Process operational requirements - Allows you to select the operational requirements of the plant in which the load is located. The following standard system options are available:

- Emergency
- Normal plant operation
- Shutdown
- Startup

You can define additional values for this list in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the load.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Typical Schematic to an Item*, page 146
- *Apply Lookup Table Data to Equipment*, page 143
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732

Load Data Tab (Other Electrical Equipment Common Properties Dialog Box)

Provides information about the mechanical power and usage requirements of the load. You can find many of these quantities on the load nameplate. An asterisk beside a property name indicates required data.

Rated power - The nameplate power output of the load. You must enter a value with appropriate units of measure for the standard that you are using. The units of measure depend on the type of load. You can specify the default project units of measure in the Data Dictionary Manager.

Note

- For harmonic filters and capacitors, the rated power units are VAR, kVAR, or MVAR.

Supply - Select the type of supply: alternating current (AC) or direct current (DC).

Number of wires - Select the number of wires used to supply power to the load.

Rated voltage - Select the voltage at which the load is designed to operate. This quantity ensures electrical compatibility between the electrical supply and the equipment. It usually appears on the load nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Number of phases - Select the number of phases in the power supply to the load. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Phase arrangement - Select the desired value to determine which of the available phases are in use for the current load, for example 2 PH + N (two phases plus neutral). This property applies for an AC supply only and the values on this list that are available for selection depend on the number of phases that you have selected.

Demand factor - Enter a value between 0 and 1 to represent the ratio of the load power requirement at a specified *operating point* to the rated power. The demand factor influences the consumed electrical power for the load.

Frequency (Hz) - Select the operating frequency of the load. This property applies for an AC supply only. You can define and customize the available frequencies in the Data Dictionary Manager.

Coincidence factors

Allows you to enter values corresponding to the percentage utilization of the load for the selected operating mode. Also known as duty factors. Type a value between 0 and 1 for the desired mode. A value of 0 corresponds to an unused load, while a value of 1 corresponds to continuous operation.

Operating mode - Select the operating mode that describes how the load is used in practice in the plant. The standard available operating modes are:

- Continuous
- Intermittent
- Standby
- Spare

Note

- The coincidence factor corresponding to the selected operating mode provides a quantitative measure of the load behavior in that mode. You can customize the available operating modes in the Data Dictionary Manager.

Power factor and efficiency

Enter the values under conditions of 100% of the rated power and at the *demand point*. The software uses these values in the **Load Summary Report**.

Power factor - Enter the ratio of *active power* to *apparent power* at 100% of the rated power and at the *demand point*. The power factor can have a value between 0 and 1.

Note

- Power factor does not apply to harmonic filters and capacitors.

Efficiency - Enter the ratio of power output to power input at 100% of the rated power and at the *demand point*. The efficiency can have a value between 0 and 1.

Related Topics
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Electrical Calculations*, page 531
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732
- *General Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 725
- *Instruments Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 731

Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)

Provides information about the load properties based on the output power requirements.

Electrical power

The software calculates the electrical power based on the load data. The values are all read-only. The software can perform the calculation at two operating points:

- **Rated** - Maximum power.
- **Consumed** - Normal operation at the operating power.

Active - Shows the quantity of electrical power in kW drawn from the network and supplied to the load that is actually converted to output power.

Reactive - Shows the quantity of reactive power in kVAR that the load draws from the network.

Apparent - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This quantity is the actual power that the network needs to generate.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on the **Load Data** tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the load draws from the supply when operating at maximum power. The full load current usually appears on the nameplate of the load. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually. When the load has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Related Topics
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Electrical Calculations*, page 531
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732
- *General Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 725

Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)

Provides optional information for control stations that control the load operation, and controls and signals for the load.

Associated control stations

Displays a list of control stations associated with the load. This information is read-only and appears either if you associated a control station with the load from the **Control Station Common Properties** dialog box.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the item where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be the electrical equipment, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the load, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the load.

Related Topics
- *Apply a Profile to an Item*, page 147
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732

Instruments Tab (Other Electrical Equipment Common Properties Dialog Box)

Displays instruments that monitor and control the operation of the equipment.

Associated instruments

Displays a list of instruments associated with the equipment. This information is read-only and appears if you associated an instrument with the equipment from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Related Topics
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732
- *General Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 725
- *Load Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 726

Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in loads that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

Note

- Once you associate the equipment with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the load feeder. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

Note

- Where the feeder source is an item of equipment other than a PDB, the source tag appears in the **Feeder equipment** box and all the properties in this section are disabled.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the load, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the load is assigned. The values in this list depend on the selected power distribution board. If you select a bus without specifying a circuit, the software assigns the bus to the load, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display a cell/bucket/ compartment to which the load is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the load is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

Note

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the load is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the load, the software assigns circuit components based on that typical circuit.

✎ Note

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the load-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the load is assigned. The software automatically assigns the cell name.

Phase 1 - Select to associate the current load with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current load with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current load with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

✎ Note

- To update the circuit components, you need to modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Feeder equipment

Where equipment other than a PDB circuit feeds the load, the software displays the item tag of the feeding equipment in this box. For example, if a transformer feeds the load, the transformer tag is displayed.

Note

- Where a feeder equipment tag appears, the properties under the **Location** section are disabled. Likewise, where the feeder source is a PDB, the source data appears under the **Location** section and this property is disabled.

Last applied typical circuit

Indicates the last typical circuit that you applied to the load, if appropriate.

Tag - Displays the tag of the last applied typical circuit.

Function - Displays the function of the last applied typical circuit. For a load, this is always **Feeder**.

Related Topics
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *General Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 725
- *Instruments Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 731
- *Load Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 726

Alternative Feeder Tab (Other Electrical Equipment Common Properties Dialog Box)

Displays the location of an alternative load feeder, if defined, in loads that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the load is assigned, if defined.

Circuit - Displays the circuit to which the load is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics
- *Controls and Signals Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 730
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 728
- *Feeder Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 732
- *General Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 725
- *Instruments Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 731
- *Load Data Tab (Other Electrical Equipment Common Properties Dialog Box)*, page 726

Static Load Common Properties Dialog Box

Sets options for defining the general properties of a static load such as a heater. Also, this dialog box defines properties that the software uses for calculating electrical loads and associating the item with the electrical supply.

Related Topics
- *Create a Project Item*, page 53
- *Create Static Electrical Equipment*, page 132
- *Duplicate an Item*, page 38
- *Edit Item Properties*, page 38

General Tab (Static Load Common Properties Dialog Box)

Sets options for general properties that identify and characterize the load. An asterisk beside a property name indicates required data.

Consumer - You can use this property for filtering items that you want to appear in the Tabular Editor or in reports. The software selects this check box by default when you create a new static load.

Item tag - Identifies the load. The tag is the name that appears beside the load in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the load. Click the ellipsis button to open a dialog box where you can modify the item tag segments that are editable.

Note

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the load, for information only.

Manufacturer - Allows you to select a manufacturer for the load. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a load model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Process equipment - Allows you to enter a description of the equipment that the load affects. If the load was retrieved from SmartPlant P&ID and included an association with an item of process equipment, the software fills this property with the name of the process equipment.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular load.

Type - Displays the electrical equipment sub-class of the load, for example, heater or harmonic filter. The value is read-only. The software uses this description in filters and reports.

Last applied profile - Indicates the last profile that you applied to the load, if appropriate. This information is read-only.

Last applied typical schematic - Indicates the last typical schematic that you applied to the load, if appropriate. This information is read-only.

Last applied lookup table - Indicates the last lookup table that you applied to the load, if appropriate. This information is read-only.

Design type - Allows you to select a standard that specifies the construction of the load. You can define and customize design types in the Data Dictionary Manager.

Mounting - Allows you to select a standard type of mounting as specified by the vendor for the type of the load or machine. You can define and customize mountings in the Data Dictionary Manager.

Process operational requirements - Allows you to select the operational requirements of the plant in which the load is located. The following standard system options are available:

- Emergency
- Normal plant operation
- Shutdown
- Startup

You can define additional values for this list in the Data Dictionary Manager.

Notes - Allows you to enter text, for example, if you want to provide additional information about the load.

Load Data Tab (Static Load Common Properties Dialog Box)

Provides information about the mechanical power and usage requirements of the load. You can find many of these quantities on the load nameplate. An asterisk beside a property name indicates required data.

Rated power - The nameplate power output of the load. You must enter a value with appropriate units of measure for the standard that you are using. The units of measure depend on the type of load. You can specify the default project units of measure in the Data Dictionary Manager.

📝 Note

- For harmonic filters and capacitors, the rated power units are VAR, kVAR, or MVAR.

Supply - Select the type of supply: alternating current (AC) or direct current (DC).

Number of wires - Select the number of wires used to supply power to the load.

Rated voltage - Select the voltage at which the load is designed to operate. This quantity ensures electrical compatibility between the electrical supply and the equipment. It usually appears on the load nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Number of phases - Select the number of phases in the power supply to the load. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Phase arrangement - Select the desired value to determine which of the available phases are in use for the current load, for example 2 PH + N (two phases plus neutral). This property applies for an AC supply only and the values on this list that are available for selection depend on the number of phases that you have selected.

Demand factor - Enter a value between 0 and 1 to represent the ratio of the load power requirement at a specified *operating point* to the rated power. The demand factor influences the consumed electrical power for the load.

Frequency (Hz) - Select the operating frequency of the load. This property applies for an AC supply only. You can define and customize the available frequencies in the Data Dictionary Manager.

Coincidence factors

Allows you to enter values corresponding to the percentage utilization of the load for the selected operating mode. Also known as duty factors. Type a value between 0 and 1 for the desired mode. A value of 0 corresponds to an unused load, while a value of 1 corresponds to continuous operation.

Operating mode - Select the operating mode that describes how the load is used in practice in the plant. The standard available operating modes are:

- Continuous
- Intermittent
- Standby
- Spare

📝 **Note**

- The coincidence factor corresponding to the selected operating mode provides a quantitative measure of the load behavior in that mode. You can customize the available operating modes in the Data Dictionary Manager.

Power factor and efficiency

Enter the values under conditions of 100% of the rated power and at the *demand point*. The software uses these values in the **Load Summary Report**.

Power factor - Enter the ratio of *active power* to *apparent power* at 100% of the rated power and at the *demand point*. The power factor can have a value between 0 and 1.

📝 **Note**

- Power factor does not apply to harmonic filters and capacitors.

Efficiency - Enter the ratio of power output to power input at 100% of the rated power and at the *demand point*. The efficiency can have a value between 0 and 1.

Related Topics
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Controls and Signals Tab (Static Load Common Properties Dialog Box)*, page 741
- *Electrical Calculations*, page 531
- *Electrical Data Tab (Static Load Common Properties Dialog Box)*, page 739
- *Feeder Data Tab (Static Load Common Properties Dialog Box)*, page 743
- *General Tab (Static Load Common Properties Dialog Box)*, page 737
- *Instruments Tab (Static Load Common Properties Dialog Box)*, page 742

Electrical Data Tab (Static Load Common Properties Dialog Box)

Provides information about the load properties based on the output power requirements.

Electrical power

The software calculates the electrical power based on the load data. The values are all read-only. The software can perform the calculation at two operating points:

- **Rated** - Maximum power.
- **Consumed** - Normal operation at the operating power.

Active - Shows the quantity of electrical power in kW drawn from the network and supplied to the load that is actually converted to output power.

Reactive - Shows the quantity of reactive power in kVAR that the load draws from the network.

Apparent - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This quantity is the actual power that the network needs to generate.

📝 Note

- For harmonic filters and capacitors, the active power is zero, therefore the reactive power is equivalent to the apparent power.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on the **Load Data** tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the load draws from the supply when operating at maximum power. The full load current usually appears on the nameplate of the load. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually. When the load has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Related Topics
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Controls and Signals Tab (Static Load Common Properties Dialog Box)*, page 741
- *Electrical Calculations*, page 531

Controls and Signals Tab (Static Load Common Properties Dialog Box)

Provides optional information for control stations that control the load operation, and controls and signals for the load.

Associated control stations

Displays a list of control stations associated with the load. This information is read-only and appears either if you associated a control station with the load from the **Control Station Common Properties** dialog box or if you applied a profile defined in the **Reference Data Explorer**.

Item Tag - Displays the control station tag for the associated control station.

Type - Displays the reference control station when you create the associated project control station by copying a reference control station from the **Reference Data Explorer**. For a control station that you created in the **Electrical Index**, no value appears in this column.

Associated I/O signals

Describes all the I/O signals that are associated with the item where the signals need monitoring or operation at the PLC / DCS.

Item Tag - Displays the signal tag for the associated I/O signal.

Equipment Type - Displays the equipment type of the item that the signal originates from, which can be a heater, a capacitor, a harmonic filter, an associated circuit, or a control station.

Description - Displays a description of the associated I/O signal.

I/O Type - Displays the signal I/O type according to whether the signal is an input or an output, and whether the signal type is analog or digital.

Loop Number - Displays the loop number, if one exists, for signal data that you retrieved from SmartPlant Instrumentation.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the load, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the load.

Related Topics
- *Apply a Set of Signals to an Item*, page 148
- *Electrical Data Tab (Static Load Common Properties Dialog Box)*, page 739
- *Feeder Data Tab (Static Load Common Properties Dialog Box)*, page 743
- *General Tab (Static Load Common Properties Dialog Box)*, page 737

Instruments Tab (Static Load Common Properties Dialog Box)

Displays instruments that monitor and control the load operation.

Associated instruments

Displays a list of instruments associated with the load. This information is read-only and appears if you associated an instrument with the load from the **Instrument Properties** dialog box on the **Associated Equipment** tab.

Item Tag - Displays the tag for the associated instrument.

Instrument Type - Displays the reference instrument when you create the associated project instrument by copying an instrument from the **Reference Data Explorer**. For an instrument that you created in the **Electrical Index**, no value appears in this column.

Description - Displays a description of the associated instrument.

I/O Type - Displays the system I/O type for the instrument according to whether the signal is an input or an output, and whether the signal type is analog or digital. This value appears if you defined a system I/O type on the **Instrument Properties** dialog box.

Related Topics
- *Controls and Signals Tab (Static Load Common Properties Dialog Box)*, page 741
- *Electrical Data Tab (Static Load Common Properties Dialog Box)*, page 739
- *Feeder Data Tab (Static Load Common Properties Dialog Box)*, page 743
- *General Tab (Static Load Common Properties Dialog Box)*, page 737
- *Load Data Tab (Static Load Common Properties Dialog Box)*, page 737

Feeder Data Tab (Static Load Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in loads that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

Note

- Once you associate the load with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the load feeder. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

📝 **Note**

- Where the feeder source is an item of equipment other than a PDB, the source tag appears in the **Feeder equipment** box and all the properties in this section are disabled.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the load, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the load is assigned. The values in this list depend on the selected power distribution board. If you select a bus without specifying a circuit, the software assigns the bus to the load, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display a cell/bucket/ compartment to which the load is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the load is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

📝 **Note**

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the load is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the load, the software assigns circuit components based on that typical circuit.

📝 **Note**

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the load-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the load is assigned. The software automatically assigns the cell name.

Phase 1 - Select to associate the current load with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current load with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current load with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

📝 Note

- To update the circuit components, you need to modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Feeder equipment

Where equipment other than a PDB circuit feeds the load, the software displays the item tag of the feeding equipment in this box. For example, if a transformer feeds the load, the transformer tag is displayed.

🖋 Note

- Where a feeder equipment tag appears, the properties under the **Location** section are disabled. Likewise, where the feeder source is a PDB, the source data appears under the **Location** section and this property is disabled.

Last applied typical circuit

Indicates the last typical circuit that you applied to the load, if appropriate.

Tag - Displays the tag of the last applied typical circuit.

Function - Displays the function of the last applied typical circuit. For a load, this is always **Feeder**.

Related Topics
- *Controls and Signals Tab (Static Load Common Properties Dialog Box)*, page 741
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Static Load Common Properties Dialog Box)*, page 739
- *General Tab (Static Load Common Properties Dialog Box)*, page 737
- *Instruments Tab (Static Load Common Properties Dialog Box)*, page 742
- *Load Data Tab (Static Load Common Properties Dialog Box)*, page 737

Alternative Feeder Tab (Static Load Common Properties Dialog Box)

Displays the location of an alternative load feeder, if defined, in loads that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the load is assigned, if defined.

Circuit - Displays the circuit to which the load is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics
- *Controls and Signals Tab (Static Load Common Properties Dialog Box)*, page 741
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Static Load Common Properties Dialog Box)*, page 739
- *Feeder Data Tab (Static Load Common Properties Dialog Box)*, page 743
- *General Tab (Static Load Common Properties Dialog Box)*, page 737
- *Instruments Tab (Static Load Common Properties Dialog Box)*, page 742
- *Load Data Tab (Static Load Common Properties Dialog Box)*, page 737

Transformer Connections and Tapping Dialog Box

Sets options for defining the types of connection (Delta, Wye, and so forth) and neutral grounding for 3-phase transformers in your project. Also allows you to specify tapping arrangements on the primary and secondary transformer windings for transformers with all types of feeds.

Related Topics
- *Define Transformer Connections*, page 137
- *Define Transformer Tapping*, page 138

Primary Tab (Transformer Connections and Tapping Dialog Box)

Allows you to specify values for connections and tapping on the primary transformer winding.

Tag - Identifies the transformer. The tag is the name that appears beside the item in the **Electrical Index**. The tag matches the *naming convention* for the item. Click the ellipsis button ⬚ to open a dialog box where you can modify the tag segments that are editable.

Connection - Allows you to select the type of connection for a 3-phase input.

Neutral grounding - Allows you to specify whether to use neutral grounding. The following standard options are available:

- **Ungrounded** - Does not require grounding.

- **Solid grounding** - Grounding is achieved by means of a cable connected directly to ground.

- **Impedance** - Grounding is achieved by means of a cable connected to ground via a resistor.

Grounding cable - Allows you to specify a grounding cable when the **Neutral grounding** value is **Solid grounding** or **Impedance**. Click the ellipsis button ⬚ to open the **Find** dialog box from which you can select a suitable grounding cable. Note that for a cable to be available for use as a grounding cable, its **Neutral Earthing** property must be set to **True**.

Grounding resistor - Allows you to specify a resistor when the **Neutral grounding** value is **Impedance**. You can use a grounding resistor with or without a grounding cable. Click the ellipsis button ⬚ to open the **Find** dialog box from which you can select a suitable resistor. You define resistors under **Static Electrical Equipment** in the **Electrical Index**. Note that on the **Find** dialog box, the software retrieves all the existing resistors regardless of whether they have or have not been associated with

transformers. Make sure that you select an appropriate resistor that has not been associated yet.

Tapping options

Tapping required - Select this check box to enable the tapping options.

Tapping - Select a tapping option. The options are for information only and do not affect the ability to enter data values for the other properties in this section.

Tapping percentage per step - Type the percentage of the total winding length to be used for each step interval for tapping.

Number of steps - Type the total number of tapping points required along the winding.

Total tapping percentage range - The calculated value of the product of the tapping percentage per step and the number of steps.

Neutral Earthing Metering Equipment

Component Type - Displays the **Component Type**.

Component Tag - Displays the **Component Tag** number.

Order - Shows the order of appearance of the metering equipment on your drawing, the lower the number the closer to the winding.

Add - Opens the **Find** dialog box and lists available PT/CT whose **Neutral Earthing** flag is **False**.

Remove - Removes the selected component

Secondary Tab (Transformer Connections and Tapping Dialog Box)

Allows you to specify values for connections and tapping on a secondary transformer winding.

Tag - Identifies the transformer. The tag is the name that appears beside the item in the **Electrical Index**. The tag matches the *naming convention* for the item. Click the ellipsis button ⊡ to open a dialog box where you can modify the tag segments that are editable.

Connection - Allows you to select the type of connection for a 3-phase output.

Neutral grounding - Allows you to specify whether to use neutral grounding. The following standard options are available:

- **Ungrounded** - Does not require grounding.
- **Solid grounding** - Grounding is achieved by means of a cable connected directly to ground.
- **Impedance** - Grounding is achieved by means of a cable connected to ground via a resistor.

Grounding cable - Allows you to specify a grounding cable when the **Neutral grounding** value is **Solid grounding** or **Impedance**. Click the ellipsis button ⊡ to open the **Find** dialog box from which you can select a suitable grounding cable. Note that for a cable to be available for use as a grounding cable, its **Neutral Earthing** property must be set to **True**.

Grounding resistor - Allows you to specify a resistor when the **Neutral grounding** value is **Impedance**. You can use a grounding resistor with or without a grounding cable. Click the ellipsis button ⊡ to open the **Find** dialog box from which you can select a suitable resistor. You define resistors under **Static Electrical Equipment** in the **Electrical Index**. Note that on the **Find** dialog box, the software retrieves all the existing resistors regardless of whether they have or have not been associated with transformers. Make sure that you select an appropriate resistor that has not been associated yet.

Connection group - Select a connection group from the list, for example Dy11, Yy0. Note that the software does not validate the connection group value with respect to the values of the other connection properties that you specify.

Phase shift (degrees) - Type a value between 0 and 360 representing the phase shift between this winding and the winding that feeds it.

Winding fed from - Select an option to specify whether the primary winding or the other secondary winding feeds the current winding (relevant to 3-winding transformers only).

Combined phase shift (degrees) - Type a value between 0 and 360 representing the phase shift between this winding and the primary winding (relevant to 3-winding transformers only).

Tapping options

Tapping required - Select this check box to enable the tapping options.

Tapping - Select a tapping option. The options are for information only and do not affect the ability to enter data values for the other properties in this section.

Tapping percentage per step - Type the percentage of the total winding length to be used for each step interval for tapping.

Number of steps - Type the total number of tapping points required along the winding.

Total tapping percentage range - The calculated value of the product of the tapping percentage per step and the number of steps.

Neutral Earthing Metering Equipment

Component Type - Displays the **Component Type**.

Component Tag - Displays the **Component Tag** number.

Order - Shows the order of appearance of the metering equipment on your drawing, the lower the number the closer to the winding.

Add - Opens the **Find** dialog box and lists available PT/CT whose **Neutral Earthing** flag is **False**.

Remove - Removes the selected component

Associate Documents Dialog Box

Enables you to associate documents with selected electrical items. Also, you can open the associated documents for editing, and you can dissociate a document if needed. An asterisk beside a property name indicates required data.

Filter by

Allows you to select a set of criteria to filter the documents that you associate with selected electrical items.

Document category - Filters the documents according to document category. This selection determines the available options in the **Document type** list. You can define and customize document categories in the Data Dictionary Manager.

Document type - Filters the document category according to document type. This selection determines the documents that the software retrieves after you click **Find** in the **Edit** group. Selecting **All types** filters the documents according to category only. You can define and customize document types in the Data Dictionary Manager. For registered reports, **select System report**.

List

Displays all the documents associated with the selected electrical items.

Add - Opens the **Find** dialog box that allows you to find a document that you want to associate with the current electrical item.

Open - Opens the document that you have selected under **List**. (Not available for internal documents.)

Remove - Removes the document that you have selected under **List**.

Properties

Displays the properties of a document that you select under **List**.

Document - Displays the name of the selected document. The value for this property appears after you click **Find** and select the document reference on the **Find** dialog box. The **Find** dialog box retrieves all the existing document references and documents belonging to the selected document category and type.

Document type - Displays the type of the selected document.

Description - Displays the description of a selected document.

Path and file name - Displays the name and the file path of the selected external file.

Define Document Reference Dialog Box

Enables you to define a reference for an external document. An asterisk beside a property name indicates required data.

Filter by

Includes the set of criteria to filter the external documents that you associate with selected electrical items.

Document category - Filters the external documents according to document category. This selection determines the external document references that the software retrieves after clicking the ellipsis button ⬚ in the **Edit** group beside **Path and file name**. You can define and customize document categories in the Data Dictionary Manager.

Document type - Filters the external document category according to document type. This selection determines the external documents that the software retrieves after clicking the ellipsis button ⬚ in the **Edit** group beside **Path and file name**. Selecting **All types** causes the software to filter the documents according to category only. You can define and customize document types in the Data Dictionary Manager.

List

Displays all the documents associated with the selected electrical items.

Open - Opens the document that you have selected under **List**.

Remove - Removes the document that you have selected under **List**.

Add - Appends a new document row with information about a document you are about to associate.

Properties

Allows you to edit the properties of a document that you select under **List**.

Document - Allows you to type the name of the selected external document.

Document type - Allows you to select the type of the selected external document.

Description - Allows you to type the description of a selected external document.

Path and file name - Displays the name and the file path of the selected external file. To navigate to the file name and path, click the ellipsis button ⬚.

Retrieve Document Dialog Box

Source XML folder - Type the path to the source folder containing the .xml files associated with the document that you want to retrieve, or click **Browse** to navigate to the desired location.

Document - Select from the list the document that you want to retrieve.

OK - Retrieves the selected document and closes this dialog box.

Batch Compare Documents Dialog Box

Allows you to compare data for the current versions of multiple selected documents with the last saved revision for each document.

Save options

Save files - Saves the selected drawings as external files.

Path - Allows you to type the file path or click **Browse** to navigate to the folder where you want to save the drawings. The software automatically displays the default path that you have set for your drawings in the Options Manager.

Save as type - Allows you to select a file format for the drawings you want to save.

Print drawings - Prints out all the drawings that you are comparing.

Print comparison reports - Prints a comparison report for each selected document.

Compare Documents Dialog Box

Allows you to compare data for the current version of the selected document with the last saved revision for that document.

Document number - Displays the document name.

Description - Displays the document description.

Revision method - Displays the revision numbering method for the current document.

Revisions

Select a revision to compare with the current version.

Archived - A check mark in this box indicates that the software archived a copy of the pertinent document revision.

Revision Number - Displays the revision number that complies with the selected revision method. After clicking **New**, the software increments the last revision by one or use the next letter in the alphabet. Also, the software allows you to type an appropriate revision number if you choose not to accept the displayed value.

Revision Date - The date when you made the revision.

Created by - The name or the initials of the person who made the revision.

Checked by - The name or the initials of the person who checked the revision.

Approved by - The name or the initials of the person who approved the revision.

Description - Displays a description of the revision.

View - Allows you to open a selected archived version of the document. The software opens archived documents in view mode only.

Compare - Compares the current version of the document with the selected revision.

Document Properties Dialog Box

Enables you to enter and manage internal document revisions, rename a document, and enter a brief document description. You can also associate items for multi-item schematics. Note that if you open this dialog box without first opening a pertinent document, all the dialog box options are in view mode only.

Revisions Tab (Document Properties Dialog Box)

Enables you to enter and manage internal document revisions. Note that if you open this dialog box without first opening a pertinent document, all the dialog box options are in view mode only.

Document number - Allows you to type the document name.

Description - Allows you to type the document description.

Multi-tag typical schematic - Allows you to select a multi-tag typical schematic when you first create a multi-tag schematic document. For a single-tag schematic, this property is disabled. A multi-tag schematic is one that includes in its definitions multi-tag blocks with header and list macros used to display the tag data. Once you associate items with the document, the software displays this property in view mode only.

Report item type - Displays the main item that the selected report template represents.

Document main item type - For certain report item types only, allows you to select an item type related to the document, for example, where the report item type is **Electrical Equipment**, you can select from this list **Cable**, **Converting Equipment**, **Motor**, and so forth. The value that you select is for information purposes only.

Revision method - Allows you to select a revision numbering method for the current document. The selected method affects the value displayed under **Revision Number** after you click **New**. If you change the revision method for a document with existing revisions, the software starts a new numbering the next time you add a revision. Note that if you select **Other**, the software does not display a revision number after clicking **New**.

Revisions

Archived - A check mark in this box indicates that the software archived a copy of the pertinent document revision.

Revision Number - Displays the revision number that complies with the selected revision method. After clicking **New**, the software increments the last revision by one or use the next letter in the alphabet. Also, the software allows you to type an appropriate revision number if you choose not to accept the displayed value.

Notes

- If you type a value that does not comply with the current revision method, the next time you add a new revision, the software starts a new numbering. For example, the selected revision method is P0, P1, P2 and the last revision is P3. If you add a new revision number RRR and then another one using the P0, P1, P2 method, the software displays the newly added revision as P0.

- If you type a value that complies with the numbering method but the value you is not sequential, the software increments the number of the next revision that you add. For example, the selected revision method is P0, P1, P2 and the last revision is P3. If you add a new revision number P1, the software increments the following revision to P2.

Revision Date - The date and time of the revision.

Created by - The name or the initials of the person who made the revision.

Checked by - The name or the initials of the person who checked the revision.

Approved by - The name or the initials of the person who approved the revision.

Description - Allows you to type a brief revision description.

View - Allows you to open a selected archived version of the document. The software opens archived documents in view mode only.

Associated Items Tab (Document Properties Dialog Box)

Enables you to associate items for multi-tag schematics. Note that if you open this dialog box without first opening a pertinent document, all the dialog box options are in view mode only.

Document number - Allows you to type the document name.

Description - Allows you to type the document description.

Multi-tag typical schematic - Allows you to select a multi-tag typical schematic when you first create the document. A multi-tag schematic is one that includes in its definitions multi-tag blocks with header and list macros used to display the tag data. Once you associate items with the document, the software displays this property in view mode only.

Associated Items

Associated Items - Displays the list of items of a particular item type that are associated with the document.

Description - Displays the item description.

Associate - Opens the **Find** dialog box to allow you to associate with the document items of a particular item type.

Dissociate - Dissociates the selected item from the document. You can only dissociate items individually.

Global Revisions Dialog Box

Enables you to assign and manage revisions to a group of documents that you select.

Activity - Allows you to select one of the following revision activities:

- **Add revision** - adds a new revision to the selected documents in accordance with the value you specify in the **Revision number** field in the **Settings** group box.. The software determines which revision method suits best the value that you typed in the **Revision number** field and adjusts the revision method accordingly. If the revision number value that you are adding conforms with method of the last existing revision, the software will continue to number the new revision accordingly. If the software cannot determine a suitable revision method for the typed value, the revision method will switch to **Other**. Also, if you do not enter a value in the **Revision number** field, the software will add a new revision using the **Other** method and will not provide a value in the **Revision number** field of the individual documents.

- **Upgrade revision** - adds a new revision by raising the last revision number of all the selected documents. The software identifies the revision numbering method of each document (P0, P1, P2; 0,1,2, A,B,C, and so forth) and raises the revision number to the next value according to the existing revision method. You cannot change the revision method but you can update all the other settings by entering new settings.

- **Delete last revision** - deletes the last revision of all the selected documents.

- **Delete all revisions** - deletes all the revisions of all the selected documents.

Revision method for items without revisions - Allows you to select a revision method for documents without revisions. This option is available only after selecting **Upgrade revision** from the **Activity** list.

Settings

Revision number - Allows you to type an appropriate revision number to be used when adding or updating revisions. This setting is not available when upgrading revisions.

Revision date - Allows you to enter the revision date.

Created by - Allows you to type the name or the initials of the person who made the revision.

Checked by - Allows you to type the name or the initials of the person who checked the revision.

Approved by - Allows you to type the name or the initials of the person who approved the revision.

Revision description - Allows you to type a brief revision description.

✎ **Note**

- The software archives all the added and upgraded revisions of the selected documents so that you can compare the data of an archived document with the current version. For more information, see Comparing Documents.

New Miscellaneous Drawing Dialog Box

Allows you to specify a new miscellaneous drawing for saving to the SmartPlant Electrical database. A miscellaneous drawing can be any drawing that was created in SmartSketch or some other CAD application. After saving this drawing to your SmartPlant Electrical database, the software allows you to associate this drawing with a SmartPlant Electrical item, and manage revisions. Note that the software does not allow you to archive miscellaneous drawings.

On clicking **OK**, the software opens the selected template and you can then insert .sym files created in SmartSketch or some other application.

Template - Click the ellipsis button ⬚ to select a drawing template (.spe file). The template determines the paper size and orientation, and the title block that appears on the drawing.

Document number - Type a document number used to identify the drawing.

Description - Allows you to type a description for the drawing if required.

Register Report Common Properties Dialog Box

Enables you to register plant reports in the SmartPlant Electrical database so that you can apply revisions to the reports in the same way as for other documents. When you register a report, it appears under **Documents** > **Registered Reports** in the **Electrical Index**.

Report template - Displays the report template associated with the report you select to register. To select a report, click the ellipsis button ⬚ and navigate to one of the Excel reports that comes shipped with SmartPlant Electrical. For the full list of shipped reports, see *Shipped Reports*, page 390.

Filter - Displays a filter, if selected, for filtering the data that you want to present in the report. To select a filter, click the ellipsis button ⬚ and choose a filter that is appropriate for the report template.

Report item type - Displays the main item that the selected report template represents.

Document main item type - For certain report item types only, allows you to select an item type related to the document, for example, where the report item type is **Electrical Equipment**, you can select from this list **Cable**, **Converting Equipment**, **Motor**, and so forth. The value that you select is for information purposes only.

Document name - Allows you to type a name for the document, which appears under the **Reports** folder.

Description - Allows you to type text description for the report if required. Press **Ctrl** + **Enter** to start a new line.

Enable document for publishing - Select to use the report for publishing data to an XML file. This option is available for reports based on item types that you can publish: cabinets, instruments, equipment (which you should filter for cabinets and instruments), and signal run. Clear the check box if you do not intend to publish data for one of these item types. Clearing the check box makes the report registration process faster.

Cabinet Common Properties Dialog Box

Sets options for defining the general properties of wiring cabinets. Also, this dialog box defines properties that the software uses for calculating electrical loads and associating the item with the electrical supply.

General Tab (Cabinet Common Properties Dialog Box)

Sets options for general properties that identify and characterize the load. An asterisk beside a property name indicates required data.

Item tag - Identifies the cabinet. The tag is the name that appears beside the cabinet in the **Electrical Index** or the **Reference Data Explorer**. In the **Reference Data Explorer**, you must enter a string in this box. In the **Electrical Index**, the item tag matches the *naming convention* for the cabinet. Click the ellipsis button ⬚ to open a dialog box where you can modify the item tag segments that are editable.

📝 **Note**

- The tag property **Comply with Naming Convention** determines the behavior of an item tag name. When the property is set to **True**, the protection masks and segment properties behave according to the settings for the item naming convention in the Options Manager. When the property is set to **False**, you can edit the tag name value freely. Whether you set this property to **True** or **False**, the software ensures compliance with uniqueness if you set the **Is Unique** parameter to **True** in the **Item Type Properties** option of the Options Manager.

Description - Allows you to type a description of the cabinet, for information only.

Manufacturer - Allows you to select a manufacturer for the cabinet. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a cabinet model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Type - Displays the type of cabinet. You can define and customize cabinet types in the Data Dictionary Manager. The software uses this description in filters and reports.

Data quality - Allows you to select a standard term that provides a qualitative measure of the reliability of the data for this particular load. For example, **Preliminary**, **Certified**, or **As Built**.

Notes - Allows you to enter text, for example, if you want to provide additional information about the load.

Electrical Data Tab (Cabinet Common Properties Dialog Box)

Provides information about the electrical requirements of the cabinet.

Cabinet rated power - The power supply that the cabinet can handle, as shown on the cabinet nameplate. You must enter a value with appropriate units of measure for the standard that you are using. The units of measure depend on the type of load. You can specify the default project units of measure in the Data Dictionary Manager.

Power factor - Enter the ratio of *active power* to *apparent power* at 100% of the rated power and at the *demand point*. The power factor can have a value between 0 and 1.

Rated active power - Shows the quantity of electrical power in kW drawn from the network and that is actually converted to output power.

Rated reactive power - Shows the quantity of reactive power in kVAR that the cabinet items draw from the network.

Rated apparent power - Shows the quantity of electrical power in kVA that represents the vector sum of the active and reactive power. This quantity is the actual power that the network needs to generate.

Supply - Select the type of supply: alternating current (AC) or direct current (DC).

Number of phases - Select the number of phases in the power supply to the load. You can select 1 or 3 phases. This property applies for an AC supply only. The value you select from this list affects the phase arrangement in this tab folder and the phase association in the **Feeder Data** tab folder.

Frequency (Hz) - Select the operating frequency of the load. This property applies for an AC supply only. You can define and customize the available frequencies in the Data Dictionary Manager.

Phase arrangement - Select a value to determine which of the available phases are in use for the current load, for example 2 PH + N (two phases plus neutral). This property applies for an AC supply only and the values on this list that are available for selection depend on the number of phases that you have selected.

Rated voltage - Select the voltage at which the load is designed to operate. This quantity ensures electrical compatibility between the electrical supply and the equipment. It usually appears on the load nameplate. You can define and customize the available rated voltages in the Data Dictionary Manager.

Number of wires - Select the number of wires used to supply power to the load.

Override FLA calculation - Select the desired option to determine whether the software uses a calculated or manually typed full load current value. If you select **Calculated**, the software calculates the full load current value based on values you enter on the **Load Data** tab; the calculated value overwrites any value that you previously entered manually. Selecting **Manual** enables data entry in the **Full load current** box, with the last calculated full load current as the starting value.

Full load current (FLA) - The electrical current, in amperes, that the load draws from the supply when operating at maximum power. The full load current usually appears on the nameplate of the load. You can select **Calculated** or **Manual** from the **Override FLA calculation** list to determine whether the software calculates this value or whether you enter a value manually. When the load has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Starting current - The electrical current, in amperes, that the cabinet uses for the associated equipment when starting. When the cabinet has an attached cable, the software copies this value to the cable and uses it as a default value for sizing the cable.

Coincidence factors

Allows you to enter values corresponding to the percentage utilization of the load for the selected operating mode. Also known as duty factors. Type a value between 0 and 1 for the desired mode. A value of 0 corresponds to an unused load, while a value of 1 corresponds to continuous operation.

Operating mode - Select the operating mode that describes how the cabinet is used in practice in the plant. The standard available operating modes are:

- Continuous
- Intermittent
- Standby
- Spare

Note

- The coincidence factor corresponding to the selected operating mode provides a quantitative measure of the cabinet behavior in that mode. You can customize the available operating modes in the Data Dictionary Manager.

Related Topics
- *Alternative Feeder Tab (Cabinet Common Properties Dialog Box)*, page 765
- *Calculate Power Requirements for an Electrical Motor*, page 130
- *Electrical Calculations*, page 531
- *Feeder Data Tab (Cabinet Common Properties Dialog Box)*, page 762
- *General Tab (Cabinet Common Properties Dialog Box)*, page 761

Feeder Data Tab (Cabinet Common Properties Dialog Box)

Provides information about the feeder and details of associated protection devices such as fuses, circuit breakers, overload relays, and so forth in cabinets that belong to your project. The values on this tab reflect the definitions that you make in the **Electrical Engineer**.

Note

- Once you associate the cabinet with a circuit, the feeder data becomes read-only. Further changes are possible only through the **Electrical Engineer**. For more information, see *Associating Electrical Equipment Common Tasks*, page 215.

Location

Enables you to specify the location of the cabinet feeder. Also, you can specify these settings by performing associations in the **Electrical Engineer**.

Note

- Where the feeder source is an item of equipment other than a PDB, the source tag appears in the **Feeder equipment** box and all the properties in this section are disabled.

Power distribution board - Allows you to specify and display the power distribution board in which the feeder is located. If you select a PDB without specifying a bus, the software assigns the PDB to the cabinet, but you can change feeder data values later if you need to.

Bus - Allows you to specify and display the bus to which the cabinet is assigned. The values in this list depend on the selected PDB number. If you select a bus without specifying a circuit, the software assigns the bus to the cabinet, but you can change feeder data values later if you need to.

Cell - Allows you to specify and display a cell/bucket/ compartment to which the cabinet is assigned. The values in this list depend on the selected bus.

Circuit - Allows you to specify and display the circuit to which the cabinet is assigned. If the circuit resides in a particular cell, the values in this list depend on the selected cell. If the circuit is linked directly to the bus, and is not linked to a cell, the values in the list depend on the selected bus.

Note

- If you select a cell without selecting an existing circuit, the software automatically creates a new circuit when you click **Apply**.

Create circuit - Select to create a new feeder circuit in the bus to which the cabinet is assigned when you click **Apply**. The software automatically assigns the circuit name. When you select this check box, the **Create cell** check box becomes active. Also, with the new circuit, if you have specified a typical circuit for the cabinet, the software assigns circuit components based on that typical circuit.

🖉 **Note**

- If you do not select this check box, then unless you select a cell, the software does not create a circuit and the cabinet-bus association does not appear in the **Electrical Engineer**. In this case, you can select the check box the next time you select this tab folder to actually create the circuit.

Create cell - Select to create a new cell in the bus to which the cabinet is assigned. The software automatically assigns the cell name.

Phase 1 - Select to associate the current cabinet with phase 1 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 1 refers to the label L1 or R, as defined by the value of the **Phase Label1** bus property.

Phase 2 - Select to associate the current load with phase 2 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 2 refers to the label L2 or S, as defined by the value of the **Phase Label2** bus property.

Phase 3 - Select to associate the current load with phase 3 of the selected bus. The availability of this option depends on the value you selected from the **Number of phases** list on the **Load Data** tab and applies only where the selected bus has an AC supply. Phase 3 refers to the label L3 or T, as defined by the value of the **Phase Label3** bus property.

Details of circuit internals - Displays details of the components in the selected circuit. The values are read-only.

🖉 **Note**

- To update the circuit components, you need to modify the selected circuit in the **Electrical Index**. For more information, refer to Related Topics.

Component Type - Displays the type of component in the circuit, such as a circuit breaker or a fuse.

Component Tag - Displays the component identifying tag, as shown in the **Electrical Index**.

Order - Determines the position of the component in relation to the bus when the circuit contains more than one component. The greater the value, the further the component is from the bus.

Show in SLD - Indicates whether the software shows a graphical representation of the component when you generate a single line diagram for the PDB.

Related Topics
- *Alternative Feeder Tab (Cabinet Common Properties Dialog Box)*, page 765
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Cabinet Common Properties Dialog Box)*, page 761
- *General Tab (Cabinet Common Properties Dialog Box)*, page 761

Alternative Feeder Tab (Cabinet Common Properties Dialog Box)

Displays the location of an alternative feeder, if defined, in cabinets that belong to your project. You specify these settings by performing associations in the **Electrical Engineer**. All properties on this tab are view-only.

Location

Power distribution board - Displays the power distribution board in which the feeder is located.

Bus - Displays the bus belonging to the power distribution board.

Cell - Displays a cell/bucket/ compartment to which the cabinet is assigned, if defined.

Circuit - Displays the circuit to which the cabinet is assigned.

Feeder equipment - Displays the item used as the alternative supply (generator, battery bank, off-site power and so forth).

Related Topics
- *Copy Circuit Components*, page 172
- *Create Circuit Components*, page 173
- *Electrical Data Tab (Cabinet Common Properties Dialog Box)*, page 761
- *Feeder Data Tab (Cabinet Common Properties Dialog Box)*, page 762
- *General Tab (Cabinet Common Properties Dialog Box)*, page 761

Control Station Common Properties Dialog Box

Enables you to associate a control station with a cable and an electrical item such as a motor, heater, capacitor, harmonic filter, resistor, heat traces, UPS, VFD, battery chargers, other converting and electrical equipment.

Item tag - Identifies the control station. The item tag appears beside the control station in the **Electrical Index**. The item tag matches the *naming convention* for the control station. Click the ellipsis button ![...] to open a dialog box where you can modify the tag segments that are editable.

Last applied typical I/O set - Indicates the last typical I/O set of signals that you applied to the control station, if appropriate. This information is read-only and only appears if you apply a typical I/O set by dragging it from the **Reference Data Explorer** or by applying options to the control station.

Associated equipment - Displays the electrical item associated with the control station.

✎ Note

- You can associate one electrical item only per control station.

Find - Opens the **Find** dialog box, which allows you to select an electrical item to associate with the control station.

Dissociate - Allows you to clear the association of the selected equipment with the control station.

Associated cables

Displays the project cables that the software associates with the control station.

(Data window) - Describes the names and cable specifications of the associated cables.

Add - Opens the **Find** dialog box, which allows you to select control, grounding, or instrumentation cables to associate with the control station.

✎ Note

- You can associate more than one cable per control station.

Remove - Removes the selected associated cable from the control station.

Related Topics
- *Apply a Set of Signals to an Item*, page 148

Project Management Table (As-Built)

The Project Management table in As-Built allows you to select a target project and to scope items for that project, and to merge items in the project back to As-Built after you have edited or created them in the project. The commands associated with the **Project Management** table are all on the **Project Management** toolbar. This table opens when you click **Window > New > Project Management** on the main menu bar.

Header	Description
(Check box)	Applies to main items only. When selected, includes those items in the action specified the **Scope**, **Clear**, and **Merge** commands. The check box in the main header row is used to select or clear all the check boxes in the column.
Main Item Tag	The tag of the main item. A main item can be a motor or some other load, converting equipment, a power source, any free cable, a PDB, or a document.
Related Item Tag	The tag of the related item which has an electrical association with the main item; for example, if the main item is a motor, related items can include cables, signals, control stations, and circuits.
Item Type	Indicates the item type of the main item or related item.
Status in project	Indicates the status of the item. For details, see *Understanding Statuses*, page 472.
Claim Mode	Indicates whether the item, if claimed, has been claimed exclusively or shared, or whether it has been released from the claim. In As-Built, this property is read-only.
New Item	Indicates whether an item has been newly-created in the project. Note that after merging the item, this field appears blank.
Item Path	Indicates the parent hierarchy of a non-unique item. For example, in the case of a circuit, the item path shows the PDB\Bus\(Cell) hierarchy.
Description	The item description, if appropriate, as it appears for the item properties.

Header	Description
Plant Group	Indicates the plant group to which the item belongs.
Deleted	Indicates whether the item has been deleted in the project since the last time it was scoped.
Result Status	Indicates the result of a major operation such as **Merge**. For example, the status **Done** appears after the software commits an operation successfully.

Project Management Toolbar

Contains the commands that you can run from the **Project Management** table in As-Built.

Batch Cable Connection Dialog Box

Allows you to connect, in batch mode, numerous project cables that belong to the same category, for example, power, control, instrumentation, and so forth. For example, you can you can terminate all the project control cables that are associated with 10hp motors.

Cable side 1 (From) / Cable side 2 (To)

These group boxes allows you to define the **From** and **To** sides of the cables you are going to connect.

Activate cable side 1 (From) / **Activate cable side 2 (To)** — Activate the options for the **From** and **To** sides of the cables you are going to connect.

Terminal strip sequence — Allows you to type a value that defines the terminal strip that is associated with the equipment that you want to connect. The value that you type corresponds to the terminal sequence in group.

First terminal — Allows you to type a value that instructs the software as to which terminal on the current terminal strip is to be connected first.

Active connection side — Allows you to select the side of the terminals to be connected to the cable end.

- **Left**— The active side

- **Right**— The opposite side

Connection type — Allows you to select a connection method for the **From** and **To** sides of the cables. You can select one of the following:

- **Continuous**— Allows you to make a connection in a sequential way by selecting a terminal that will be wired first. The software then automatically wires the rest of the terminals in a sequential order.

- An existing **custom** connection type — Allows you to make your connections based on an existing reference connection type of a selected cable. You start by selecting a terminal to be wired first and the software then connects the rest of the terminals according to the connection type. If the connection type that you require is not on the list, create it in the **Reference Data Explorer**. For details, see *Define a Connection Type*, page 337.

Cable connection details

Select — Allows you to select the cables to be included in the batch connection.

Cable Tag — Displays the names of the cables that you can select for the batch connection.

Cable Category — Displays the category of the cables, for example Control, Power, and so forth.

Cable Formation — Displays the formation of the cables, for example 3x1.5 mm2.

From Equipment — Displays the name of the equipment associated with the **From** side of the current cable.

To Equipment — Displays the name of the equipment associated with the **To** side of the current cable.

From Terminal Strip — Displays the name of the terminal strip on the equipment that is associated with the **From** side of the current cable.

To Terminal Strip — Displays the name of the terminal strip on the equipment that is associated with the **To** side of the current cable.

Select all — Selects all the cables displayed in the data window.

Cable Connection Type Selection Dialog Box

Allows you to select a cable connection type that you can use to wire project items using the custom connection method in a wiring drawing or when connecting cables in batch mode.

Cable — Displays the name of the cable that you selected to terminate.

Item — Displays the name of the wiring item you selected in the current wiring drawing or in the **Batch Cable Connection** dialog box.

Cable connection details

This data window contains all the cable sets of the current cable and connection types for each cable set.

- **Select**— Allows you to select an appropriate cable set in a multi-set cable.

- **Cable/Set**— Displays the name of the cable or cable set you are selecting for termination.

- **Connection Type** — Allows you to select an appropriate connection type for the current cable termination. If the connection type that you require is not on the list, create it in the **Reference Data Explorer**. For details, see *Define a Connection Type*, page 337.

Connect — Effects the termination of the terminal you selected in the current wiring drawing.

Connection Type Dialog Box

Allows you to define a connection type that facilitates rapid cable set connections. You define a connection type that contains a pattern by which SmartPlant Electrical connects conductors to terminals on a terminal strip.

Connection type — Allows you to type a connection type name. Click the ellipsis button [...] to open a dialog box where you can modify the connection type segments.

Description — Allows you to type a description connection type, for information only. Click the ellipsis button [...] to open a dialog box where you can type longer text for the description.

Note

- When typing the description in the dialog box, to begin a new line, press **Ctrl** + **Enter**.

Skip between sets — Allows you to enter the number of terminals that you want to skip between the first terminal of each set.

Conductors

This data window contains all the conductors of the selected cable set, showing their names and configurations. The number of rows that you should add has to match the number of conductors that will be connected. The software does not connect the leftover conductors that do not have connection type definitions in the **Conductors** data window.

Conductor — Displays a specific conductor in a cable set. The displayed **Wire00x** values represent the conductor names and are used to illustrate the current connection type configuration. This field is not editable.

Skip — Allows you to type the number of terminals to skip before connecting this conductor in the set. The skip refers to the number of terminals that will be left unconnected between consecutive cable sets. Enter the number of terminals to be skipped according to your needs. In most cases, this value is incremented for successive conductors.

Land Side — Allows you to set the side of the terminal to which you want to connect each conductor.

- **A** — The active side of the terminal. In the environment of a wiring drawing, the **A** side is the active side of the terminal that you want to connect. It can be its left or right side, depending on which terminal side you clicked after you moved the cursor over the connection points.

- **O** — The opposite side of the terminal. In the environment of a wiring drawing, the **O** side is the opposite side of the terminal that you want to connect. It can be its left or right side, depending on which terminal side you clicked after you moved the cursor over the connection points.

Add — Adds a new row to the **Conductors** data window where you can define a new conductor connection. The number of rows must match the number of conductors you are going to connect. The software does not connect the leftover conductors that do not have connection type definitions in the **Conductors** data window.

Remove — Removes the selected row from the **Conductors** data window.

Terminal Strip Configuration Dialog Box

Allows you to define a reference terminal strip configuration that you can apply to a terminal strip whose properties you are editing in the **Electrical Index**.

Note that SmartPlant Electrical creates terminal strip configurations based on the number of terminals that you enter and terminal strip pattern that you define.

Configuration — Allows you to type a name for the current reference terminal strip configuration.

Description — Allows you to type a description of the current reference terminal strip configuration.

Configuration properties

Total number of terminals — Allows you to enter a number of terminals to be created on the current terminal strip. Note that this number should always be higher than the

First terminal — Allows you to type or select a starting number of the first terminal on the current terminal strip.

Terminal strip pattern

This data window displays the pattern of terminal numbering on the terminal strip based on your **Total number of terminals** value. The pattern reflects the number of rows (terminals) that will constitute the basic repeating numbering (naming) pattern.

Sequence — Displays the sequential number of a terminal.

Prefix — Allows you to type a prefix for the terminal if you require.

Incremented — Increments the terminal number within the pattern.

Numbered — Makes the number of the terminal appear between the prefix and the suffix.

Suffix — Allows you to type a suffix for the terminal if you require.

Add — Adds a new row to the terminal strip pattern. The maximum number of rows that you can add cannot exceed the total number of terminals that you enter.

Remove — Removes the highlighted row from the terminal strip pattern.

Terminal Strip Common Properties Dialog Box

Allows you to define the properties of a terminal strip and set its terminal configuration.

Item tag — Allows you to type a terminal strip name. Click the ellipsis button [...] to open a dialog box where you can modify the terminal strip tag segments.

Description — Allows you to type a description of the terminal strip, for information only. Click the ellipsis button [...] to open a dialog box where you can type longer text for the description.

✎ Note

- When typing the description in the dialog box, to begin a new line, press **Ctrl** + **Enter**.

Terminal strip configuration

Allows you to apply a reference terminal strip configuration to the current terminal strip. Also, you can define a terminal strip configuration on the fly. The software creates the terminal strips and the terminals based on the configuration you use. However, the software does not retain the configuration that you create on the fly. To save a configuration for later use, define a reference terminal configuration in the **Reference Data Explorer**. For details, see *Define a Reference Terminal Strip Configuration*, page 333.

Total number of terminals — Allows you to enter a number of terminals to be created on the current terminal strip.

First terminal — Allows you to enter a starting number for the first terminal on the current terminal strip.

Find — Opens the **Find** dialog box that allows you to find an appropriate reference terminal strip configuration that you want to apply to the current terminal strip.

Terminal strip pattern

This data window displays the pattern of terminal numbering on the terminal strip based on your **Total number of terminals** value. The pattern reflects the number of rows (terminals) that will constitute the basic repeating numbering (naming) pattern.

Sequence — Displays the sequential number of a terminal.

Prefix — Allows you to type a prefix for the terminal if you require.

Incremented — Increments the terminal number within the pattern.

Numbered — Makes the number of the terminal appear between the prefix and the suffix.

Suffix — Allows you to type a suffix for the terminal if you require.

Add — Adds a new row to the terminal strip pattern.

Remove — Removes the highlighted row from the terminal strip pattern.

Terminal Common Properties Dialog Box

Enables you to define the properties of a terminal.

General Tab (Terminal Common Properties Dialog Box)

Allows you to define the general properties of a terminal.

Item tag — Allows you to type a terminal name. Click the ellipsis button [...] to open a dialog box where you can modify the terminal strip tag segments.

Equipment — Displays the name of the associated electrical equipment.

Description — Allows you to type a description of the terminal, for information only. Click the ellipsis button [...] to open a dialog box where you can type longer text for the description.

🖉 **Note**

- When typing the description in the dialog box, to begin a new line, press **Ctrl** + **Enter**.

Terminal strip — Displays the terminal strip that is associated with the current terminal.

Type — Allows you to select a type for the current terminal. You can define and customize types in Data Dictionary Manager.

Terminal color — Allows you to select a color for the current terminal. You can define and customize terminal colors in Data Dictionary Manager.

Sequence — Allows you to enter the sequential number of the current terminal on its terminal strip.

Procurement Tab (Terminal Common Properties Dialog Box)

Enables you to enter information related to ordering a specific terminal for your inventory.

Manufacturer - Allows you to select a manufacturer. You can define and customize manufacturers in the Data Dictionary Manager.

Model - Allows you to select a terminal model. The available models are specific to the manufacturer. You can select a model without selecting a manufacturer. The software then selects the appropriate manufacturer automatically according to the selected model. You can define and customize models in the Data Dictionary Manager.

Store part number - Allows you to enter a reference number for the terminal in your inventory. This read-only value comes from the reference cable used as the basis for this cable.

Custom Symbols Dialog Box

This dialog box allows you to associate a customized symbol file with each available document type (drawing), for each specific item you use in SmartPlant Electrical.

Item Tag — Displays the item tag selected for associating a symbol with.

Document Type — Allows you to select the document type for the item tag that you want to associate.

Symbol — Allows you to select the custom symbol you want to associate with the document type and item tag. Click ⋯, to view the custom symbols in the default **Symbols** folder and its sub-folders. The default Symbol folder is defined under **Options** > **Locations** in the SmartPlant Electrical Options Manager.

Add — Adds a new row for defining a document and symbol association. The **Add** button is deactivated when no more rows can be added.

Remove — Removes the selected row from the item tag.

Table Properties Dialog Box

Enables you to select the item types, filters, and layouts for a Tabular Editor view. This dialog box appears when you click **Window > New > Table** on the main toolbar or when you right-click **Table Properties** on the Tabular Editor view shortcut menu.

Advanced - Displays the **Advanced Table Properties** dialog box, where you can define and save filters or layouts for a Tabular Editor view. You can specify the default filter and layout for a certain item type also. Specify brief and bulk properties in this dialog box, too.

Filter - Lists all of the filters that have been saved for the selected item type.

Item Type - Lists all the item types that you can select - for example, equipment.

Layout - Lists all of the layouts that have been saved for the selected item type.

Related Topics
- *Define a Filter for a Tabular Editor View*, page 101
- *Define a New Layout for a Tabular Editor View*, page 102
- *Table Properties Command (Edit Menu)*, page 557

Advanced Table Properties Dialog Box

Allows you to define, select, and save filters or layouts for a Tabular Editor view. This dialog box opens when you click **Advanced** on the **Table Properties** dialog box, which opens when you click **Window > New > Table** on the main toolbar or when you right-click **Table Properties** on the Tabular Editor view shortcut menu.

Filter Tab (Advanced Table Properties Dialog Box)

Define, select, and save a filter for a Tabular Editor view.

Base filter - Contains details of saved filters.

Name - Displays the name of a saved filter.

Default - Indicates if the named filter is the default filter for the item type selected on the **Table Properties** dialog box. You can designate a filter as the default by checking this box.

Browse - Opens the **Select Filter** dialog box, which allows you to select a base filter.

Match all - Items matching ALL of the filtering criteria pass through the filter.

Match any - Items matching any one or more of the filtering criteria pass through the filter. **Match any** is the default matching method.

Add - Places a new entry at the end of the existing definition list. Enables the fields in the **Edit** group so that you can edit the new entry.

Delete - Removes the selected criterion from the definition list. This button is available only when a row is selected in the definition list.

Edit - Allows you to define or edit a single line of filter definition criteria.

Property - Displays a list of all properties for a certain item type. Examples of properties include **Equipment Type**, **Instrument Type**, and **Estimated Length**. You can define or modify criteria by selecting a property, an operator, and a value.

Operator - Select the relationship between the property and its value.

Value - Lists values associated with the property you selected in the **Property** list. If no properties are listed, you can specify one. You can type a percent sign (**%**) as a wildcard character to find multiple characters, or type an underscore (**_**) as a wildcard character for a single character. Do not use an asterisk in the **Value** box.

Layout Tab (Advanced Table Properties Dialog Box)

Define, select, and save a layout for a Tabular Editor view.

Name - Displays the name of a saved layout. You can enter a new layout name and then click **Save** to save the layout for re-use. Or, you can select any of the layouts in the list.

Default - Indicates whether the named layout is the default layout for the item type on the **Table Properties** dialog box. You can designate a layout as the default by selecting this box and then clicking **Save**.

Usage - Lists the property display types associated with the **Properties** window. Options include a blank value, **Brief**, **Bulk**, and **Brief/Bulk**. The selected item type and display attributes define what appears in the **Properties** window when you select the **Show Brief Properties**, **Copy Bulk Properties**, or **Paste Bulk Properties** commands from the **Properties** window toolbar.

💡 **Tip**

- You can have only Brief and one Bulk layout per item type. The last layout that you define as Brief or Bulk is set as such and replaces the previous one.

Delete (Name) - Removes the named layout.

Save - Saves the data in this dialog box as a named layout. Layout names must be unique.

Definition - Displays all defined criteria associated with the selected layout. This group contains two list views. In order to add to or modify either list, you must select a line in the list and then define or edit the property in the **Edit** group. The buttons to the right act upon the active list view.

Display Property - Lists the columns that appear in the Tabular Editor. To edit an entry, select it and modify the property in the **Edit** group.

Caption (Definition) - Displays the caption for each column in the Tabular Editor.

Sort Property - Lists the properties that apply to the item type specified in the **Table Properties** dialog box.

Order - Lists the sort orders **ascending** and **descending**. You can select one.

Type - Lists the sort types **numeric** and **alphanumeric**. You select one.

Add - Adds a new entry at the end of the existing entries. Enables the fields in the **Edit** group so that you can edit the new entry.

Insert - Places a new entry above the currently selected entry. Enables the fields in the **Edit** group so that you can edit the new entry.

Delete (Definition) - Removes the currently selected entry.

Up - Moves the currently selected entry up one line.

Down - Moves the currently selected entry down one line.

Edit - Allows you to edit a single entry selected in either the **Display Property** or **Sort Property** list views. If you select an entry in the **Display Property** view, the **Edit** group allows you to edit the property and the caption. If you select an entry in the **Sort Property** view, the **Edit** group allows you to edit the property, order and type.

Property - Displays a list of all the properties that apply to the item type defined in the **Table Properties** dialog box. If you select an entry in the **Display Property** view, you can specify the property and its caption. If you select an entry in the **Sort Property** view, you can specify the property, its order, and its type.

Caption (Edit) - Displays the caption for each column in the Tabular Editor.

Select Filter Dialog Box

Allows you to select a filter for filtering the data for a particular item type to display in a Tabular Editor view or in a report. Using this dialog box, you can create a new filter or modify an existing filter. To display this dialog box, you can:

- Click **Browse** on the **Filter** tab of the **Advanced Table Properties** dialog box while you are choosing your item type for displaying in a Tabular Editor view.

- Click **Browse** on the **Filter** tab of the **Define Report Items** dialog box while you are choosing your report item type.

New - Displays the **New Filter** dialog box, which allows you to specify a new simple or new compound filter.

Properties - Allows you to edit the properties of the selected filter. This button displays the **Filter Properties** dialog box or the **Compound Filter Properties** dialog box, depending on your selection in the filter list. If you create a compound filter, you select the individual simple filters that comprise a compound filter and click **Properties** to view the **Filter Properties** dialog box for those simple filters.

Advanced Publish Options Dialog Box

Allows you to specify the type of files that you want to search for when you look for documents to publish. This dialog box opens when you click **Advanced** on the **Publish** dialog box.

Document type — Indicates the types of documents the software considers when selecting the documents to publish.

Document Properties Dialog Box

Provides details about a new or existing document selected for publishing. This dialog box opens when you click **File System** on the **Publish** dialog box.

Selected file — Displays the name of the file that you selected on the **Select File** dialog box.

New document — Indicates that this document has not been published previously.

Published previously — Indicates that the file has already been published on at least one previous occasion.

Document category — Allows you to select a category to assign to the document.

Document type — Allows you to select the type of the document. The options that appear in this list are determined by the selection you make in the **Document category** box.

Document subtype — Lists the subtype of the document. If applicable, select one. The options that appear in this list are determined by the selection you make in the **Document type** box.

Name — Allows you to enter the name of the file as it is known in the SmartPlant Foundation database.

Descriptions — Allows you to enter a brief description of the file. This description appears later to help you recognize the file.

Title — Allows you to enter the official title of the document.

Note

- An asterisk ***** next to a box indicates that the information is mandatory.

Related Topics
- *Find Documents to Publish Command (SmartPlant Menu)*, page 568

Find Documents to Publish Dialog Box

Allows you to search for documents that have been updated since they were last published or that were previously published but no longer exist in your authoring tool (SmartPlant P&ID or SmartPlant Instrumentation, for instance). The dialog box opens when you click **SmartPlant** > **Find Documents to Publish** on the main menu bar.

Last search performed — Displays the date when the files were last searched. The information that appears in the lists on this dialog box were found on this date and time.

Update — Performs a new search for documents to publish.

Document types searched — Indicates what types of files were considered when the last search was conducted.

Select documents to publish — Displays a list of files that were either updated since they were last published or files that have not yet been published. For each file, this list displays the name and type of the file and the date that it was last published. If the file has not been published, the **Last Published** box for the document displays the value **New**.

Select documents to terminate — Displays a list of all the files that were previously published but were subsequently removed from the project. For each file, this list displays the name and type of the file and the date that the document was last published.

Select All — Selects all the files in the **Select documents to publish** list.

Clear All — Cancels the selection of all the files in the **Select documents to publish** list.

Related Topics
- *Find Documents to Publish Command (SmartPlant Menu)*, page 568

Publish Dialog Box

Displays a list of documents to publish. This dialog box opens when you click **SmartPlant** > **Publish** on the main menu bar.

Selected documents — Displays a list of the documents selected for publishing. You can populate this list by clicking the buttons in the **Add** area of this dialog box. For each document, this list provides the name, the type of the document, the application which last published the document, the revision and version numbers, the revision schedule, and the date that the document was last published.

Engineering Tool — Opens a tool-specific dialog box that allows you to select documents to add to the **Selected documents** list.

File System — Opens the standard Microsoft **Select File** dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file in this dialog box, you open the **Document Properties** dialog box, and you can specify information about the file, such as whether it is a new file or was previously published; the category, type, and subtype of the document; and the name, description, and title of the document.

Find — Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list.

Advanced — Opens the **Advanced Publish Options** dialog box, which allows you to search for files that are of a specific type and that have been updated since they were last published.

Remove — Removes the selected file or files from the **Selected documents** list.

Contract — If applicable, opens the **Contract** dialog box. This command is available only if it is defined by your project implementation team.

Batch publish — Indicates that the system publishes the selected documents in batch mode (that is, in the background). You are notified by e-mail when the operation is complete. If this options is not selected, the documents begin to be published as soon as you click **OK**.

📝 Note

- You may select rows that are in white on this dialog box. Rows that are gray are provided for viewing purposes only and cannot be selected.

Retrieve Dialog Box

Allows you to specify options for retrieving information. The dialog box opens when you click **SmartPlant** > **Retrieve** on the main menu bar.

Document type — Indicates the type of documents you can retrieve and changes the list view so that it displays only that document type.

Show — Allows you to indicate whether you want to view a list of all of the latest documents available or of only those documents that need to be retrieved.

Documents to retrieve — Displays a list of the documents to be retrieved. For each document, this list provides the name, the type of document, the revision and version numbers, the last retrieval date, the status of the document, and the **Revision** option. By using the **Revision** option, you can indicate whether you want to retrieve the document as it was published or with the latest data.

Select All — Selects all the files in the **Documents to retrieve** list.

Clear All — Cancels the selection of documents in the **Documents to retrieve** list.

Batch retrieve - Indicates that the system retrieves the selected documents in *batch* mode. You are notified by e-mail when the operation is complete. If this option is not selected, the documents are retrieved as soon as you click **OK**.

To Do List Dialog Box

Provides the ability to manage tasks that need to be run within a SmartPlant Electrical database to update the plant after retrieving information. The commands associated with the **To Do List** are all on the **To Do List** toolbar. This dialog box opens when you click **SmartPlant** > **To Do List** on the main menu bar.

Task List - Populates automatically when you retrieve a document. This list provides information about each task, including, for example, the type of the task, the item type, the item tag affected by the task, and so forth. By clicking the heading of any of these columns, you can sort the list by the values in that column. Alternately clicking the same column heading toggles the sort between ascending and descending order. Double-clicking a task opens the **Task Properties** dialog box for that task.

To Do List Toolbar

Contains the commands that you can run from the **To Do List** dialog box.

Defer Task Command

SmartPlant > To Do List > Defer Task

Toggles the status of the selected tasks between **Open** and **Deferred** . This command is available only when the selected task or tasks have either **Open**, **Error** , or **Deferred** status.

Delete Task Command

SmartPlant > To Do List > Delete Task

Allows you to delete the selected task from the **To Do List**. This command is available only when one or more tasks are selected. When you use this command in the **To Do List**, the software moves the tasks to the deleted tasks list. When you use this to delete permanently one or more tasks in the deleted tasks list, a confirmation dialog box appears to verify the deletion.

View Deleted Tasks Command

Displays a list of tasks that have been deleted from the **To Do List**. This command toggles this feature on and off. If you want to completely delete tasks from the list of deleted tasks, you can select them and click **Delete Task** ✕ again on the **To Do List** toolbar.

Related Topics
- *Display Deleted Tasks on the To Do List*, page 529
- *Remove a Task from the To Do List*, page 529
- *Using the To Do List: An Overview*, page 525

Properties Command (To Do List Dialog Box)

Displays the **Task Properties** dialog box for the selected task. This dialog box shows the detailed properties of the task. This command is available only when a single task is selected.

Refresh Command

SmartPlant > **To Do List** > **Refresh**

Updates the information that is displayed in the **To Do List**. Since only one **To Do List** is generated per plant, multiple users can be working in the **To Do List** at the same time, and so the information can change at any time. You can use the **Refresh** command to stay up-to-date.

Run Task Command

SmartPlant > **To Do List** > **Run Task**

Runs the *task* that you select from the list on the **To Do List** dialog box. This command is available when an executable task is selected and the status of the selected task is **Open** or **Error**. If multiple tasks are selected, this command is available only if each of the tasks meets these criteria.

When a task runs, the status of the task and the **Run Date** and **Run By** properties are updated. If the task runs successfully, the status is updated to **Completed** ✓. If there is a problem, the status changes to **Error** ⓘ, and a timestamp and error message are added to the **Notes** area.

Task Properties Dialog Box

Provides information about tasks that are defined in the task list of the **To Do List** dialog box. The type of task that you selected from the **To Do List** determines the information that appears on this dialog box. There are three basic types of tasks: **Create**, **Update**, and **Delete**. You must select a task from the **To Do List** in order to open this dialog box. This dialog box opens when you click **Properties** 🖼 on the **To Do List** dialog box toolbar.

General Tab (Task Properties Dialog Box)

Displays properties that are common to all types of tasks.

Task

Displays the status and type of task, and a description of what happens when the task is run.

Name - Provides a descriptive string that helps you identify the purpose of the task. This value is not a unique identifier for the task.

Status - Indicates the level of completion of the task. The following choices are available: **TaskOpen**, **TaskCompleted** ✓, **TaskDeferred** ▬, and **Error** ⓘ. When a task is created, the status is set to **TaskOpen**, which does not have an identifying icon in the **Task** list on the **To Do List**.

Type - Describes the task type, for example, **Manual Create**, **Manual Update**, or **Manual Delete**.

Description - Allows you to type a description for the task, for example, of what happens in the database when the task is run.

Source

Provides general information about the task source.

Engineering Tool - Displays the name of the application that published the document.

Document - Displays the name of the document from which this task originated, for example, an instrument index from SmartPlant Instrumentation.

Item tag - Displays the item tag of the source object from which this task originated. This information is read-only.

Destination

Provides general information about the item tag affected by the task.

Item tag - Displays the item tag of the item to be processed in the SmartPlant Electrical database. For **Update** and **Delete** tasks, this box displays the item tag of an existing item. For **Create** tasks, this box displays the item tag of the item that the software is going to create. This information is read-only.

Related Topics
- *Modify To Do List Task Properties*, page 528
- *Properties Command (To Do List Dialog Box)*, page 787

Details Tab (Task Properties Dialog Box)

Provides information that is common to all types of tasks and that concerns the source and destination of the information being imported as part of the task.

Item type - The item type that the software assigns to the item when entering the property values in the SmartPlant Electrical database. Examples of item types are **SignalRun**, **Instrument**, and **Cabinet**.

Item properties

Displays the task data properties and enables you to select which properties to include when you run the task. For an **Update** task, only those properties for which the values have changed are shown.

Property - The name of the item property as it appears in SmartPlant Electrical.

New Value - The new value of the item property in the SmartPlant Electrical database as indicated in the retrieved document. This column only appears if the task type is **Create** or **Update**.

Old Value - The current value of the item property in the SmartPlant Electrical database. This column only appears if the task type is **Update**.

History Tab (Task Properties Dialog Box)

Provides information this is common to all types of tasks and that concerns the history of the task. This information includes the user names and dates for the creation of an item, modification of an item, and so forth. The information on this tab is read-only.

Created

Displays creation details for the task.

By - Displays the user name of the task creator.

Date - Displays the date and time that the task was created.

Modified

Displays modify details for the task, other than for **Create** tasks.

By - Displays the user name that last ran, modified, or deleted the task.

Date - Displays the date and time that the task was last run, modified, or deleted.

Run

Displays details for the task when last run.

By - Displays the user name that last ran the task.

Date - Displays the date and time that the task was last run.

Deleted

Displays details for a deleted task. Note that this refers to deletion of the task itself, not of the item that the task affects.

By - Displays the user name that removed the task.

Date - Displays the date and time that the task was removed.

Notes Tab (Task Properties Dialog Box)

Allows you to enter freeform text about the task. You can also enter hyperlinks to other documents for more extensive information. Information about the execution of the task, such as the success of the procedure, is added to the end of any existing notes for a task.

Log Files Dialog Box

Allows you to select from a list of categories and display an error log file for the selected category. You can generate an error log file for any batch operation in SmartPlant Electrical as well as for bulk copying and pasting of values in the **Properties** window. Note that you need to activate the error log file generation on the **Preferences** dialog box before you attempt to generate any error log file.

Log file type - Select the category for which you want to view the log file. Note the functionality of the following two error log files:

- Comprehensive Error Log File - contains all errors of all types.
- Miscellaneous Error Log File - contains errors that do not fit the description of any of the available log file types.

Clear Log - Clears the contents of the log for the selected log file type.

Show

Allows you to specify which types of errors appear in the log files.

All log file data - Displays data and rule inconsistency errors.

Rule inconsistencies only - Displays rule inconsistency errors only.

All log file data excluding rule inconsistencies - Displays data errors only.

Show log file data for last operation only - Select to display errors for the last performed operation relating to the specified log file. Clear to display accumulated errors for successive operations.

Open - Opens a log file of the selected category in Microsoft Excel. After generating the log file in Excel, you switch back to the **Log File Selection** dialog box and select another category. Clicking **Open** displays the log in the same Excel workbook but in a different worksheet.

Close - Closes this dialog box.

Rule Inconsistency Statistics Dialog Box

Enables you to view a summary of rule violations that may have occurred for the selected items. You can also display the log file in Excel format if you require.

Summary

Total number of items - Shows the total number of items that you selected in the **Electrical Index** or Tabular Editor.

Inconsistency Type - Indicate the type of rule inconsistency, if any, that the software found. The following levels are indicated: **No inconsistency**, **Logged**, **Warning**, and **Rejected**, according to the options specified in the **Rule Manager**.

Number of Items - Indicates the number of selected items that the software found with inconsistencies corresponding to the indicated inconsistency type.

Cancel - Cancels the processing while the software is checking items for rule inconsistencies. When the software has finished checking all the selected items, this command changes to **Close**.

Show Log - Displays an Excel file showing details of rule inconsistencies that the software found for the selected items.

Rule Inconsistencies Dialog Box

Enables you to view rule inconsistencies for operations related to the selected items.

Data Window

Displays the rules with details of inconsistencies and the action to be taken.

Rule Name - The name of the rule as defined by the user. The same rule name can appear more than once if several inconsistencies exist for that rule.

Inconsistencies - Indicates the way in which the data is inconsistent with the rule for the specified item. The following example specifies an inconsistency for a rule whereby a 100hp motor must have a rated voltage of at least 400V:

```
Motor=100hp(M-A-1000):Rated Voltage(220V)>=(400V)
```

Where the components of the formula are as follows:

Component	Value
Operand 1	Motor=100hp
Item tag	(M-A-1000)
Property	Rated Voltage
Actual database value	(1000V)
Operator	>=
Operand 2 property / value	(400V)

In this case, the actual rated voltage for motor M-A-1000 is 220V, and so this value represents a rule inconsistency.

Action - Specifies the action that is required where there is an inconsistency. The actions can be:

- **Log**— Records the rule inconsistency to a log file without notifying you. You can access the log file from SmartPlant Electrical.

- **Warn**— Prompts you to reject or accept the operation when a rule inconsistency is found. This action gives you flexibility in making an appropriate decision.

- **Reject**— Automatically rejects the operation that you performed and notifies you about the rule inconsistency. This action prevents you from accepting any operation when the software detects an inconsistency.

Reject Operation - Rejects the operation on the item or relationship for which the software finds rule inconsistencies. In the case of rule inconsistencies for which the action is specified as **Warn**, the software does not save the data. Note that this command does not appear when one or more rule inconsistencies exist for which the action is specified as **Reject**.

Accept Inconsistencies - Performs the operation, even though data values or relationships exist that are inconsistent with the rules. In the case of rule inconsistencies for which the action is specified as **Warn**, the software accepts the latest data values and saves them in the database. Note that this command does not appear when one or more rule inconsistencies exist for which the action is specified as **Reject**.

Close - This command is available for closing the dialog box when one or more rule inconsistencies exist for which the action is specified as **Reject**.

Show Rule - Opens the **Rule Properties** dialog box with read-only information showing details of the selected rule.

Rule Properties Dialog Box

Enables you to view the properties of the rules that appear on the **Rule Inconsistencies** dialog box.

Rule properties include the following settings:

- Rule name and description
- Rule components (that is, item types or filters created for certain item types in Filter Manager)
- Logical expressions which determine rule consistency criteria
- Action associated with a logical expression (an action applies whenever the user operation in SmartPlant Electrical does not comply with the logical expression)

General Tab (Rule Properties Dialog Box)

Enables you to view the rule name, description, or priority.

Rule name — Defines the name of the rule.

Rule description — Identifies the purpose or function of your rule.

Disable rule — Not in use. Note that only those rules appear for which this check box was not selected in the **Rule Manager**.

Priority — Determines which rule applies first when rules conflict with each other. Values lie in the range from 1 to 100, where 1 represents the lowest priority and 100 represents the highest. When a user of SmartPlant Electrical performs an operation that involves items to which several rules apply, the software first applies the rules with the highest priority.

In a case of rule inconsistency, if a specific logical expression in the rule is associated with the **Reject** action, the software rejects the operation automatically and does not apply any other rules that has lower priorities. Likewise, if a logical expression is associated with the **Warn** action, and the user rejects the operation, the software does not apply any other rules that has lower priorities.

Note

- A priority applies to an entire rule, not to a particular logical expression.

Rule Components Tab (Rule Properties Dialog Box)

Enables you to view the rule type — relation rule or single component rule — together with the rule components. Rule components can be either filters created in Filter Manager or item types available in SmartPlant Electrical (except for reference data item types). A filter specifies an extensive class of item types to which the rule applies. A relation rule consists of a pair of rule components. A single component rule has only one rule component.

Component 1

Displays the first rule component and set or changes the rule type.

Filter or item type — Displays the first rule component.

Single component — Indicates the rule type. When this check box is selected, the software uses a single component rule for the component specified in the **Filter or item Type** box under **Component 1**. A single component rule enables you to evaluate and validate consistency of the properties or property values of a specific item tag.

Component 2

Available only when the rule type is a relation rule. Specifies a second rule component, which the software can display as an operand in logical relations in the **Consistency** tab.

Filter or item type — Displays the second rule component.

Note

- The order of the rule components is not important. Relation rules are be bi-directional. If your rule components are different, and in the **Consistency** tab you create a logical expression using both components as operands, the rule applies to any of the components involved in user operations in SmartPlant Electrical.

Consistency Tab (Rule Properties Dialog Box)

Defines consistency check criteria and enables you to view logical expressions that exist for the rule.

Whenever the software detects a rule violation, it performs the action assigned to the specific logical expression. Depending on an action, the software can record the inconsistency to a log file, reject the user operation automatically, or prompt the user to accept or reject the operation.

Each row is the tab is a specific logical expression that consists of various parameters for creating a relationship between a pair of rule components or between one rule component and its property or a specific value. A user operation that does not satisfy all of the consistency criteria is considered a rule violation.

Consistency criteria

Displays the consistency criteria which consist of one or more logical expressions and an action associated with specific expressions. A logical expression is a relationship created between two operands by means of a logical operator.

A logical expression consists of the left and right parts separated by an operator:

- Left part — Must always contain Operand 1 and its property, where Operand 1 represents a rule component.

- Right part — Can contain either a rule component as an Operand 2 and its property or the **Value** operand and a value specified for the property of Operand 1.

Operand 1 — Displays a rule component specified in the **Rule Components** tab. If the rule type is single component, the cell displays the rule component specified on the **Rule Components** tab under **Component 1**.

Property — Lists appropriate properties for the rule component that appears in the **Operand 1** list.

Operator — Displays the operator for the logical relation. For details about operator descriptions, see Operator Descriptions and Rule Examples.

Operand 2 — Displays a rule component specified in the **Rule Components** tab or the **Value** operand of a property selected in the **Property** column. For a rule component, this column displays the relationship between properties of rule components. For the **Value** operand, this column displays the relationship between the rule component property and the property value.

Property/Value — Depending on the selection in the **Operand 2** column, lists properties or values. If the Operand 2 column contains a rule component, this column displays a property of a specific item type or filter. If the Operand 2 column contains the **Value** operand, this column displays the value for the property defined in the **Property** column. Depending on the operator, a value can be free text, a number of units of measure separated by commas or semi colons, or a value selected from the list.

Action — Displays the action for the software to perform in case of inconsistency. In SmartPlant Electrical, the software applies an action only when the software detects a rule inconsistency (that is, whenever a user violates the logical expression with which you associate the action). The software records rule inconsistency details into a log file which you can generate from SmartPlant Electrical.

The actions are:

- **Log**— Records the rule inconsistency to the log file without notifying you.

- **Warn**— Prompts you to reject or accept the operation when a rule inconsistency is found. This action gives you flexibility in making an appropriate decision.

- **Reject**— Rejects user operation automatically and notifies the user about the rule inconsistency. This action prevents you from accepting the operation when the software detects an inconsistency.

Related Topics
- *Operator Descriptions and Rule Examples*, page 124
- *Rule Properties Dialog Box*, page 793

SmartPlant Electrical Glossary

active power - The quantity of electrical power (in W, kW, or MW) supplied to a consumer that is capable of producing actual work. See also: rated active load, consumed active load.

actual length - A derived value that SmartPlant Electrical uses for cable sizing, cable drum assignment, and so forth. If a cable has a value for Estimated length, but no value for Design length, then the Estimated length is also the Actual length. If there is a value for Design length only, then this is also used as the value for Actual length.

ANSI - American National Standards Institute.

apparent power - The quantity of electrical power (in VA, kVA, or MVA) that represents the vector sum of the active load and reactive load for a motor. This quantity is the actual power that the electricity supply company needs to generate.

As-Built - The operational plant that is used as the source of data for your projects.

asynchronous speed - The maximum attainable rate of rotation of a motor shaft, in rpm. This quantity is always less than the synchronous speed because of energy losses due to the magnetic field, pole architecture, and so forth.

average power - The average electrical power consumption for a consumer, used to calculate overall electrical load requirements. The average load is calculated by multiplying the rated active load by the operating factor.

block - A block is the basic drawing unit of a schematic. Each block graphically represents the functionality of an item or a related group of items. Each block must belong to a block type, which you can use to classify the blocks according to the items they represent; for example, a motor and its cables, or a circuit block with its bus, cell, and PDB. The block also specifies the drawing file that contains the graphical information for the items.

brake power - The minimum mechanical power required to make the motor shaft rotate and drive the pump or process equipment attached to the motor.

bucket - See cell.

buffer - A temporary data store used for setting aside items before you decide whether to scope those items.

bus riser, bus tier - See riser circuit.

busbars - Busbars are used in place of standard cables to interconnect electrical equipment requiring high current. They consist of strips of copper, usually arranged in groups of three or four. Each set of busbars operates at a specified rated voltage.

cell - A removable compartment in a power distribution board that contains circuitry.

claim - Selection of an item in a project so that it can be edited in that project.

claim mode - Indicates whether a claimed item is available for all available projects (shared claim) or only for the selected project (exclusive claim).

coincidence factor - The fraction of the maximum possible utilization of the motor for a specified operating mode. The software uses this factor when calculating the total load consumption of a power distribution board or electrical system.

compartment - A section of a power distribution board that contains the required circuitry.

consumed active load - The quantity of electrical power used by a consumer when operating at the brake power.

consumer - See load.

control station - A local or remote panel containing a low-voltage circuit, used to control the operation of a motor via a control circuit.

converting equipment - Equipment used to convert one level or type of electric property to another. Converting equipment includes transformers, battery chargers, UPS units, and variable-frequency drives (VFDs).

coupler circuit - A circuit that connects two buses in a power distribution board for the purpose of creating a bus tie. This circuit belongs to the bus that serves as the power source of the coupled bus in the PDB.

cubicle - See cell.

demand factor - A quantity expressing the ratio of brake power to motor rated power. The demand factor can have values between 0 and 1.

demand point - The point where the motor operates at its brake power.

disconnect electrical equipment - Equipment used to disconnect a load from the power supply. Disconnect electrical equipment includes switches, starters, overload relays, fuses, circuit breakers, and protection relays.

electrical distribution network - The arrangement of buses, cables, and distribution boards used to bring electrical power to the loads.

distribution board - A box divided into panels, used to distribute the power supply to the various loads, and to transform the voltage as required. See also: power distribution board.

distribution equipment - Equipment used to distribute the power supply among several loads. Distribution equipment includes power distribution boards (PDBs), motor control centers (MCCs), disconnect electrical equipment, cables, and panels.

efficiency - A quantity expressing the ratio of useful power output to power input. Efficiency can have values between 0 and 1.

electrical distribution network - The arrangement of buses, cables, and distribution boards used to bring electrical power to the loads.

enclosure - The physical housing of a piece of electrical equipment used to protect the equipment from its environment.

enclosure protection - The level of protection afforded to a motor from its environment. For example, the operational environment of the motor may require that the enclosure is sealed or explosion-proof.

EPC - Engineering Procurement Contractor: a company that has responsibility to design and erect a plant.

feeder - A cable or circuit that distributes power from a power distribution board to other equipment.

frame size - A US standard that defines the physical and mechanical properties and dimensions of a motor.

full load amperes (FLA), full load current (FLC) - The electrical current drawn by a motor when operating at maximum load. This quantity is required for sizing cables that supply power to the motor.

full load current - See full load amperes.

IEC - International Electrotechnical Commission, a European electrical standard.

incomer - A cable or circuit that supplies power to a power distribution board.

item status in project - Describes the status of a particular item in the project. Possible statuses are: Scoped, Claimed, Completed, or Merged.

load (consumer) - An electrical device, such as a motor or heater, that draws power from the electrical distribution network.

load center - See power distribution board.

load factor - See demand factor.

locked rotor current (LRC) - The initial electrical current required to start rotation of a motor. Depending on the motor, this quantity is typically 400 - 1000% larger than the full load current of the motor. Usually, the motor nameplate specifies the LRC/FLA ratio, and this ratio is multiplied by the motor FLA to calculate the LRC value.

macro - A macro is a string attached to a graphic block that represents the item type and one of its properties. You can assign one or more macros to a block. When you generate a schematic drawing, the software reads and resolves the macros so that it can retrieve the value of the property from the database and display it in the drawing.

main item - A term used to describe almost any item added in As-Built that can be a focal item with related items. Main items can include loads, converting equipment, power sources, free cables, PDBs, and documents. You can only scope main items if they are included in a plant group that has been scoped.

merge - The action of moving items whose data was modified in a project back to the As-Built schema.

minimum required power - Under the IEC Standard, the minimum mechanical power that the motor shaft is required to supply when operational.

miscellaneous drawings - Drawings that you create based on a template, to which you can add symbols. You can revise miscellaneous drawings, as you can any other SmartPlant Electrical document, and associate it with any item in your plant.

motor control center (MCC) - A power distribution board providing the necessary electrical power for motor loads.

motor rated power (MRP) - The maximum mechanical power that a motor shaft is required to supply. This quantity is usually marked on the motor nameplate, and is used in calculating the electrical power requirements for the motor.

motor speed - The rate of rotation of the motor shaft, in rpm. Motor speeds are characterized by two standard quantities: the synchronous speed and the asynchronous speed.

naming convention - The naming convention is a means of ensuring a consistent structure and maintaining uniqueness of tags for items of the same type. Once you define a naming convention for an item type such as motors, the software tags any new items of that type using the mask and structure of the naming convention. The naming convention is a combination of properties, text strings, and separators that makes up the tag structure in the Electrical Index. You define naming conventions in the Options Manager.

NEC - National Electric Code, a US electrical standard.

operating factor - A quantity expressing the ratio of the load power requirement at the operating point to the rated power. The operating factor can have values between 0 and 1.

operating mode - A description of how the motor is used in practice in the plant. The operating mode can be: continuous, intermittent, normal, standby, or spare, and a quantitative measure of the motor behavior in each mode is provided by the coincidence factor.

Operating owner - A company that owns and operates a plant.

operating point - The point where the motor operates under normal working conditions.

panel circuit - A circuit that can be created in a local panel or a junction box.

particular power - Under the IEC Standard, the mechanical power that a motor shaft is required to supply when operating under a specified set of conditions, for example, once a year.

power distribution board (PDB) - A distribution board containing components of the electrical distribution network at the supply level.

power factor (cos f) - The ratio between active load and apparent load. The power factor can have values between 0 and 1.

profile - A profile is an assembly of various typical items that you can use as a template for creating project items. For example, you can create a motor profile that consists of the following reference items: a motor, one or more cables, a control station, a circuit with protection devices, and schematic.

project - A separate electrical schema within a plant used to claim items for updating when performing routine maintenance or modernization (revamp) for that plant. To facilitate plant modernization, you can create a number of projects within a single plant. Within a particular plant, the projects and As-Built have a common reference schema and data dictionary.

project status - Describes the status the project as a whole. Possible statuses are: Active, Completed, Merged, or Cancelled.

rated active load - The maximum quantity of electrical power used by a consumer when operating at the rated power.

rated power - The maximum output power that a load can produce. See also: motor rated power.

rated voltage - The voltage at which a piece of equipment is designed to operate. This quantity is required to ensure electrical compatibility between the electrical supply and the equipment.

reactive power - The component of the supplied electrical power (in VAR, kVAr, or MVAR) that the load uses to provide reactive energy.

reference plant group - A plant group containing reference data, options settings, naming conventions, and so forth, that you can use as a source for copying. Usually, a reference plant group does not belong to your current plant structure. You can access a reference plant group and view naming conventions only from Options Manager.

related item - An item that has an electrical relationship to a main item and which the software scopes automatically when the main item is scoped. Related items may include control stations, associated cables, circuits, and signals. Some scoped items may belong to unscoped plant groups.

relation rule - A rule in which consistency criteria apply to two specific components. Relation rules ensure that correct relationships are created when you create a new relationship or modify properties or values of related items in SmartPlant Electrical.

resistive load - A load that has resistive characteristics.

riser circuit - A circuit that connects two buses in a power distribution board for the purpose of creating a bus tie. This circuit belongs to the bus that serves as the receiver of backup power from the feeder bus in the PDB.

rotor - The rotating part of a motor or generator.

run-up time - The time it takes a motor to reach its operating speed after switching on the power.

schematic - A graphical representation of the wiring of a load, its feeder circuit and its controls, which includes specific data for that particular load.

scope - In Engineering Manager, the scope defines the extent of the items that you are able to view and subsequently claim, based on plant group levels of the project. In SmartPlant Electrical, any item that you scope in As-Built becomes available for viewing in the project. When you select and scope main items, the software includes certain related items in the scope.

segregation level - A term used to group cables that share a common electrical signal and can therefore be run along the same routing path. You use segregation levels to distinguish between different groups of cables that if routed together could produce electrical or magnetic interference. Examples of such groups include high or low voltage cables, cables of different types or sources, or cables associated with different applications or systems.

shaft - The part of a motor or generator that drives the attached equipment.

signal - An item type that is used for mediating between SmartPlant Electrical and SmartPlant Instrumentation. You can associate a signal with a load, a control station, or a circuit for the purpose of publishing data to SmartPlant Instrumentation. You can perform wiring connections in SmartPlant Instrumentation and retrieve the host information back to SmartPlant Electrical, where it appears as read-only properties of the signal. SmartPlant Electrical can then use the values of those properties to display I/O assignment information in a schematic.

single component rule - A rule in which consistency criteria apply to one component only. Single component rules validate consistency of the properties or property values of a specific item tag.

single line diagram (SLD) - A schematic one-line diagram representing the electrical distribution network.

static load - Equipment that does not rotate; for example, heaters, lighting equipment, other machines are static loads.

stator - The stationary part of a piece of rotating electrical equipment, such as a motor or generator.

strip - A termination entity used for making wiring connections. A strip is always at the lowest level in the hierarchy of distribution equipment.

switch board - See power distribution board.

switchgear - See power distribution board.

synchronous speed - The maximum theoretical rate of rotation of a machine shaft, in rpm, assuming no energy losses due to the magnetic field, pole architecture, and so forth. See also: asynchronous speed.

Te stall time - The time it takes a motor to come to rest after switching off the power.

Index